Multiframe

Windows Version 16

User Manual

© Bentley Systems, Incorporated 2013

License & Copyright

Multiframe Program & User Manual
© 2013 Bentley Systems, Incorporated

Table of Contents

License & Copyright iii
Table of Contents v
About this Manual 1
Chapter 1 Getting Started 3
    Installing Multiframe 3
    Learning Multiframe 3
Chapter 2 Using Multiframe 5
    Techniques 5
        Summary of Mouse Techniques 5
        Summary of Keyboard Techniques 6
    Using Views 7
        View Directions 7
        Rotating a 3D View 8
        3D Rotation 8
        Zoom, Pan, Shrink and Size To Fit 8
        Clipping and Masking 12
        Rendering 16
        Selecting Joints and Members 18
        Selecting Load Panels 18
        Selecting Patches 19
        Selecting Plates 20
        Selecting Patch Openings 20
        Select Patch Edges 21
        Select Opening Edges 22
        Symbols 23
        Legends 24
        Centre of Gravity 27
        DXF Background 28
    Creating a Structure 29
        Drawing 29
        Setting the Size 29
        Drawing Grid 30
        Structural Grid 30
        Drawing Depth 33
        Snapping to Joints and Members 33
        Dynamic Line Constraints 34
        Drawing Settings 35
        Setting the Origin 36
        Adding a Member 36
        Adding a Load Panel 37
        Adding a Patch 39
        Editing a Patch 41
        Add N-sided Patches & Openings 43
        3D & Curved Patch Support 45
        Edit Patch Openings 48
        Delete Patch Openings 49
        Edit Patch/Opening Edges 49
        Extrude Rider Bar 49
        Delete Patch/Opening Edges 51
        Meshing a Patch 51
        Patch Mesh Patterns 52

Displaying a Patch or Plate 53
Attach Member/Patch to Patch 54
Detach Member/Patch to Patch 55
Align Member to Patch or Load Panel 56
Automatically Align and Offset Member to Patch/Load Panel 57
Member Offset Connections 57
Add Rounded Corner Rectangular Opening 58
Convert Opening to Patch 59
Opening Duplicate 60
Opening Mirror 61
Opening Move 62
Opening Rotate 63
Patch/Opening Edge Extrude 64
Generate N-sided Patch & Opening 65
Patch Split 67
Patch Merge 69
Auto-generate Patch Stiffeners 70
Auto-generate Patch Hot Spots 72
Connections Between Members 74
Selections 74
Subdividing a Member 76
Converting a Member into an Arc 78
Merge Members 79
Intersect Members 79
Deleting a Member 80
Automatic Generation 81
Duplicate 87
Cylindrical Coordinates 89
Spherical Coordinates 91
Moving a Joint 91
Typing Joint Coordinates 92
Adding Joints 93
Aligning Joints 93
Moving a Group of Joints 94
Moving a Member 95
Disconnecting Members 96
Resizing a Member 97
Rescaling the Structure 97
Rotating Members 98
Extruding Beams or Columns 99
Mirroring Members 99
Shearing Members 100
Editing Coordinates Numerically 101
Joint and Member Numbers 103
Joint and Member Labels 104
Restraints 104
Joint Restraint 105
Springs 106
Editing Restraints Numerically 107
Grouping 107
Linking Joints Or Master-Slave 110
Joint Mass 112
Member Masses 112
Section Orientation 113

Automatically Align and Offset Member to Patch 114
Section Properties 115
Adding a Standard Section....115
Adding a Custom Section 117
Removing a Section 119
Editing a Section....119
Section Type 120
Materials 120
Member Materials....121
Patch Orthotropic Materials....121
Adding a Material 122
Editing and Deleting Materials 123
Working with the Sections Library....123
Joint Type 124
Joint Orientation 125
Member Releases....126
Member Types 127
Member Offsets 128
Member End Springs 129
Member Modelling 131
Shear Area....131
Applying Loads....132
Joint Load 133
Joint Moment 133
Prescribed Displacement....134
Global Distributed Load 135
Local Distributed Load 137
Global Panel Load 138
Local Panel Load 138
Global Patch Distributed Load....139
Local Patch Distributed Load 141
Global Patch Point Load 141
Local Patch Point Load....142
Patch Edge Loads....143
Global Point Load....145
Local Point Load....146
Global Moment 147
Local Moment....148
Thermal Load....149
Editing Loads Numerically 150
Self Weight 151
Load Cases....154
Dynamic Loads 158
Sea Motion Load Case 163
Wind Load Case....166
Water Current Load Case....172
Buoyancy Load Case 176
Hydrostatic Load Case 178
Evaluating Expressions....181
Analysis....183
Linear Analysis 184
Nonlinear Analysis 184
Patch Stresses....187
Lifting Analysis 189

Buckling Analysis....190
Modal Analysis....192
Time History Analysis....193
Batch Analysis....194
Viewing Results....195
Result Window....195
Joint Displacements....196
Joint Reactions....196
Member Actions....197
Maximum Actions....198
Member Stresses....198
Maximum Stresses....199
Member Details....199
Spring Actions....201
End Spring Actions....201
Patch Internal Forces....201
Patch Stresses....201
Nonlinear Results....203
Natural Frequencies....203
Member Buckling....203
Time History Results:....203
Viewing Time History Envelope Cases....205
Plot Window....206
Structure Diagrams....207
Stresses....214
Member Diagrams....215
Plate Diagrams....215
Rendering Patch Results....218
Add Patch Cross Section....219
Deleting a Cross Section....220
Envelope Case Plots....222
Joint Reactions....223
Joint Displacements....224
Buckling Results....224
Modal Results....224
Calculations....225
Calculation Sheet....225
Pre-defined CalcSheet Variables....226
Section Properties Variables....227
Saving Calculations....228
Printing....228
Page Setup....228
Setting up the Printer....228
Printing Reports....228
Printing Diagrams....229
Print Window....230
Data Exchange....230
File Import....230
RealDWG....232
Import Construction Lines....234
File Export....239
Graphics....244
Table Data....245
Animations....245

Chapter 3 Multiframe Reference 247
Windows 247
Frame Window 247
Data Window 247
Load Window 247
Result Window 247
Plot Window 247
CalcSheet Window 247
Report Window 247
Toolbars 248
File Toolbar 248
Generate Toolbar 248
Drawing Toolbar 248
Geometry Toolbar 249
Member Toolbar 249
Joint Toolbar 250
View Toolbar 250
Actions Toolbar 250
Load Case Toolbar 251
Load Toolbar 251
Formatting Toolbar 252
Symbols Toolbar 252
Windows Toolbar 253
View3D Toolbar 253
Render Toolbar 253
Group Toolbar 253
Clipping Toolbar 254
Load Panel Toolbar 254
Load Panel Symbols Toolbar 254
Patch Toolbar 255
Patch Symbols Toolbar 255
Menus 255
Patch Stresses Submenu 257
File Menu 257
Import Submenu 259
Export Submenu 259
Edit Menu 261
Sections Submenu 265
Materials Submenu 265
View Menu 265
Size To Fit Submenu 267
Clipping Submenu 268
Masking Submenu 269
Current View Submenu 269
Toolbar Submenu 270
Select Menu 272
Geometry Menu 273
Group Menu 278
Frame Menu 278
Load Menu 281
Display Menu 283
Data Submenu 286
Results Submenu 288
Actions Submenu 289

Stresses Submenu 290
Patch Internal Forces Submenu 291
Patch Stresses Submenu....292
Case Menu 292
Add Case Submenu....293
Analyse Menu 293
Time Menu....294
Window Menu 294
Help Menu 295
Chapter 4 Multiframe Analysis 297
Method of Analysis 297
Matrix Stiffness Method 297
Axes and Sign Convention....297
Member Actions 299
Modal Analysis 300
Capacity 302
Nonlinear Analysis 303
Plate Analysis 304
References 308
Appendix A Troubleshooting 309
Troubleshooting 309
Appendix B Analysing Trusses 315
Analysing Trusses 315
Appendix C Section Map File Format....317
Section Map Format 317
Appendix D Text File Format 319
Text File Format....319
Multiframe Text Example File 322
Appendix E Using Spreadsheets With Multiframe 327
Spreadsheets with Multiframe....327
Appendix F Quality Assurance 335
Quality Assurance 335
Index 337

About this Manual

This manual is about Multiframe, a software system for structural analysis and design.

Chapter 1 Getting Started
Directs you to sources for installing and learning to use Multiframe.

Chapter 2 Using Multiframe
This chapter includes step-by-step instructions in using the program. This will explain most of the tasks you will carry out in defining, analysing and designing a structure.

Chapter 3 Multiframe Reference
Gives an overview of the operations of Multiframe and a summary of the commands used.

Chapter 4 Multiframe Analysis
Discusses the numerical methods used by Multiframe. It is important for you to understand these methods and their limitations before using Multiframe for structural analysis and design. The chapter ends with a summary of the capabilities and limitations of Multiframe.

This manual describes the three versions of Multiframe, 2D, 3D and 4D. Where appropriate the manual will indicate features that are only available in the 3D and 4D versions.

Chapter 1 Getting Started

This chapter briefly describes where to find the relevant resources to

- **Installing Multiframe**
- **Learning Multiframe**

Installing Multiframe

Instructions on installing and starting Multiframe can be found in the Installation Guide included in the package you received with the installation CD and copy protection device. Alternatively, you can also consult the installation guide online: http://www.formsys.com/installation.

Learning Multiframe

The best way to learn to use Multiframe is going through the "Learning Multiframe " tutorials. These take you through all the basics of working with Multiframe from starting the application to modelling a simple 2D or 3D structure. The tutorials are supported by videos. These videos can be installed by installing "Learning Multiframe" from the installation CD or after downloading the installer from the web. Alternatively, the entire training, including videos, can be viewed online.

Go to the Learning Multiframe page on our website for the installer download and / or start the tutorials online: http://www.formsys.com/mflearning.

Within Multiframe, the Learning Multiframe training can be accessed from:

- **Help menu in Multiframe**
- **Start menu | Programs | Multiframe | Help (only when Learning Multiframe has been installed)**

Both new users as well as experienced Multiframe users will benefit from working through this training document.

Continue reading:

- **Chapter 2 Using Multiframe; series of step-by-step instructions on specific tasks in Multiframe.**
- **Chapter 3 Multiframe Reference; lists all menu commands available in Multiframe.**
- **Chapter 4 Multiframe Analysis; explains the matrix calculation methods used in Multiframe including sign conventions etc.**

Chapter 2 Using Multiframe

After doing Learning Multiframe tutorials (see Chapter 1 Getting Started for more information) you should now have a basic knowledge of some of the features of structural analysis and design using Multiframe. This chapter presents a series of step-by-step instructions to the tasks covered in Learning Multiframe as well as other procedures you will want to know about.

The chapter begins with a summary of basic computer skills, and this is followed by a description of the tasks involved in analysing and designing a structure using Multiframe. These tasks fall into three general categories; Creating the structure, specifying the loads and interpreting the results of analysis. The first three sections of this chapter reflect these categories. This is followed by detailed explanations of doing design calculations, printing and saving data and transferring data to other programs.

- Techniques
- Using Views
- Creating a Structure
- Applying Loads
- Analysis
- Viewing Results
- Printing
- Data Exchange

Techniques

There are several techniques you can use in Multiframe to make modelling structures fast and intuitive. In this section you will learn about the different mouse and keyboard techniques available in Multiframe.

- Summary of Mouse Techniques
- Summary of Keyboard Techniques

Summary of Mouse Techniques

You will use the following mouse techniques to do just about all of the tasks in this chapter.

- **Click to select or activate something**
- **Press to cause a continuous action**
- **Drag to select, choose from a menu or move something**
- **Shift-Click to select or to extend or reduce a selection**
- **Ctrl-Click to select or to extend or reduce a selection**
- **Double click to get information about an object**

To Click

Position the pointer on what you want to select or activate
Press and quickly release the mouse button

To Press

Position the pointer on something

Without moving the mouse, press and hold down the mouse button

The effects of pressing continue as long as the mouse button is held down. Pressing on a scroll arrow results in continuous scrolling. Pressing on a menu title pulls down the menu and keeps it down until you release the mouse button.

To Drag

Position the pointer on something
Press and hold down the mouse button and move the mouse
Release the mouse button

To Shift-Click

Shift-click is used to extend or reduce the selection of joints and members
Hold down the shift key and click on the joints or members you wish to add to the selection or which you wish to remove from the selection

To Ctrl-Click

Ctrl-Click is used to extend or reduce the selection of joints and members
Hold down the Ctrl key and click on the joints or members you wish to add to the selection or which you wish to remove from the selection

To double click

Double click is used to get information about a joint or member in the Frame window
Point to the item you wish to double click and then click twice quickly in succession without moving the mouse.

Summary of Keyboard Techniques

Tab

You can use the Tab key to move horizontally within a table or to move from one field in a dialog to the next.

Enter

The Enter key can be used to confirm the entry of numbers into a table and is the same as clicking OK in a dialog.

Arrow Keys

The ↑ ← → ↓ keys may be used to move the selection in their respective directions in the Data or Result tables or in tables in dialog boxes.

Delete - Backspace

The Delete or Backspace key may be used to delete the current selection. If nothing is selected and you are typing text or numbers, it will delete the character to the left of the blinking cursor.

Ctrl

The Ctrl key may be held down while typing another key to choose a command from a menu without using the mouse. Menu items that have a key to the right of the name may be chosen in this way. For example, to choose Undo from the Edit Menu you could hold down the Ctrl key and type Z.

Shift or Ctrl

You can hold down the Shift or Ctrl keys while clicking on something to add it to the current selection or remove it from the selection if it is already selected.

Holding down the shift or Ctrl while drawing a member, dragging a member or dragging a joint will constrain the movement to be horizontal, vertical or at a 45 degree angle.

Home

Takes you to the top of the table in the Data and Result windows.

F2 function key

Shortcut key to execute a Linear Analysis.

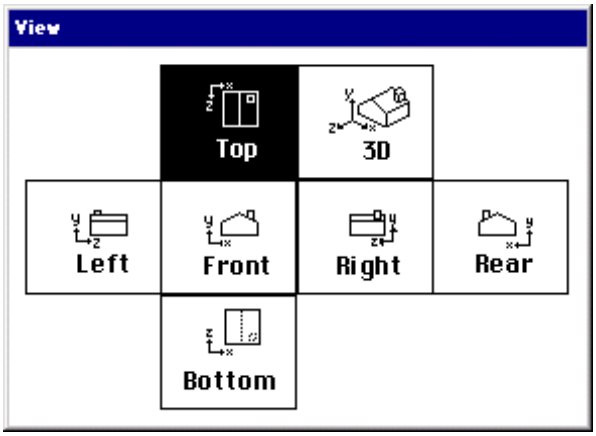
Using Views

There are a number of tools in Multiframe to help you view your frame. This section describes different utilities to display your structural model

- View Directions
- Rotating a 3D View
- Zoom, Pan, Shrink and Size To Fit
- Clipping and Masking
- Rendering
- Selecting Joints and Members
- Selecting Load Panels
- Selecting Patches
- Selecting Plates
- Symbols
- Legend

View Directions

The View button in the bottom left hand corner of the Frame, Load and Plot windows can be used to display a range of two and three-dimensional views of a structure. Each window has its own view and is controlled separately by its view button.

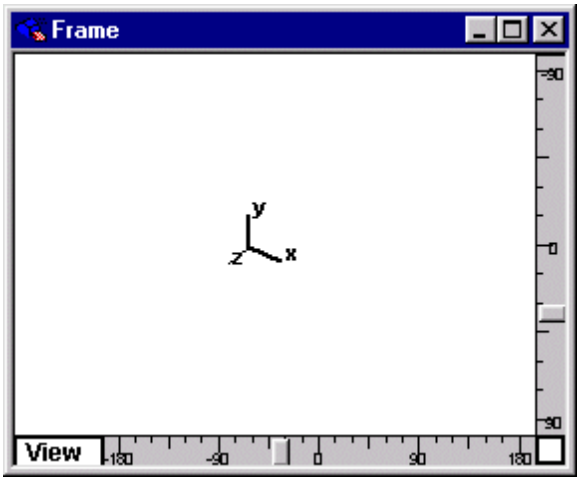


You can change the view by clicking on the View button and then clicking on the icon of the view you require. You can also change the view by using the View3D toolbar.

The view may also be changed via the Current View submenu in the View menu.

Rotating a 3D View

When you have a 3D view displayed in the Frame, Load or Plot windows, you can control the angle of view by using the rotation controls at the bottom and right sides of the window. The axes indicator indicates the current angle of view.



The control at the bottom controls the rotation about the y axis while the control on the right hand side controls the rotation about the x axis. You can click in the control to move the angle of rotation to the position of the mouse. If you hold down the mouse button after clicking in the control, you can rotate the structure back and forth until you have the desired angle of rotation. If your structure is too large to draw rapidly as you rotate it, Multiframe will draw a partial outline of the frame.

3D Rotation

There is a Rotate command in the View menu and a corresponding Rotate button on the View toolkbar. This allows for arbitrary 3D rotation of the current view in the 3D views of the Frame, Load or Plot window. The short cut to access this command is Shift-MiddleButton click (same as Autocad). The 3D rotation uses the “virtual trackball” found in most CAD systems.

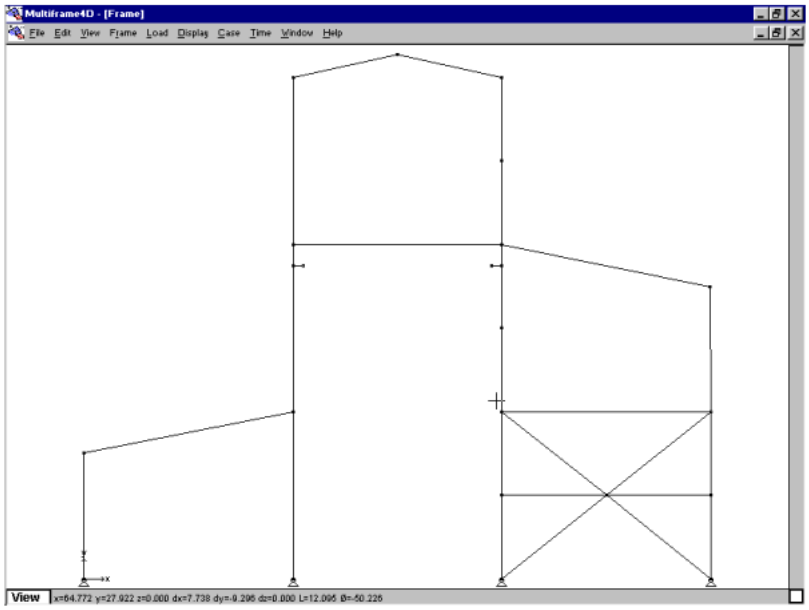
Zoom, Pan, Shrink and Size To Fit

The Zoom, Pan, Shrink and Size To Fit commands in the View Menu may be used to control the scale of the graphics displayed in the Frame, Load and Plot windows. Each view within a window has its own scale and centre of interest. This means you can have a close-up plan view and a far away 3D view in the same window and simply switch from one to another using the View button. You can also select these commands from the View toolbar.

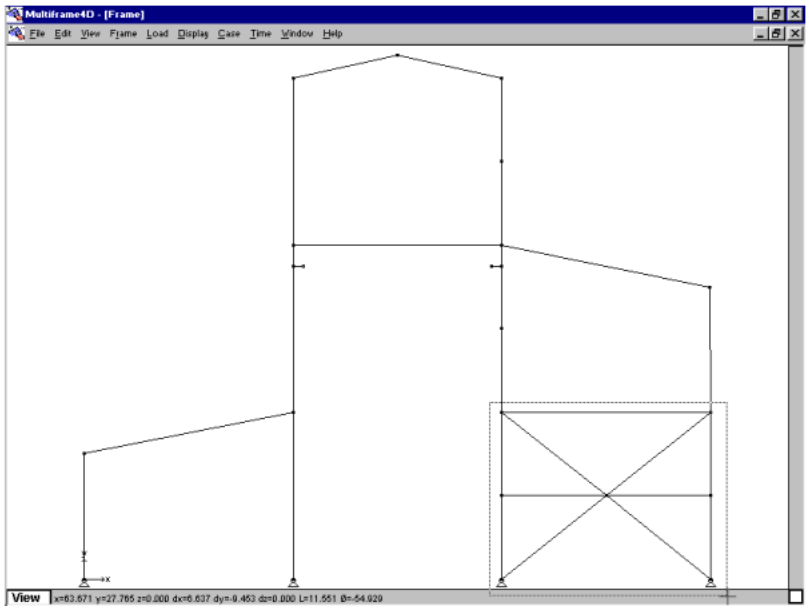
Zoom

Zoom allows you to increase the size of the drawing in the front window.

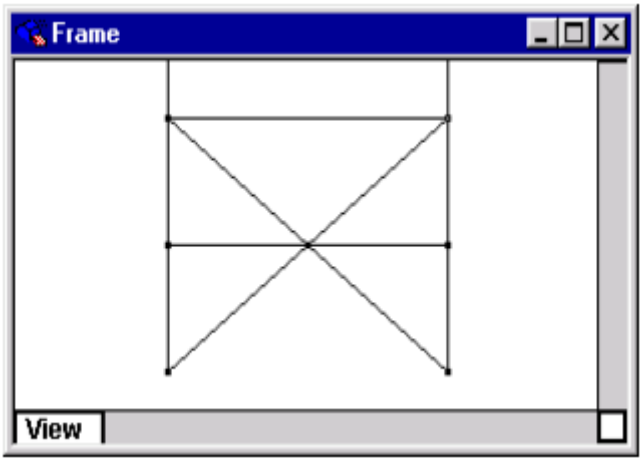
- **Choose Zoom from the View Menu**
- **Move the pointer to the top left hand corner of the area you wish to view in close detail**



- **Drag a rectangle down and to the right which encloses the area of interest and release the mouse button**



The window's contents will be re-drawn to display the part of the structure contained in the rectangle you have drawn.



If the mouse you are using has a wheel it can be used to dynamically zoom the view.

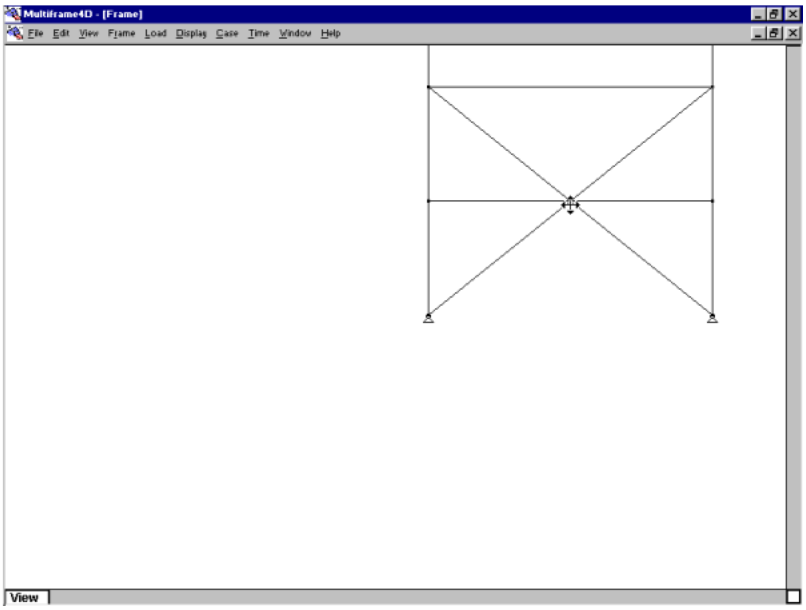
Pan

Pan allows you to shift the display of the structure within the window upwards, downwards, to the left or right.

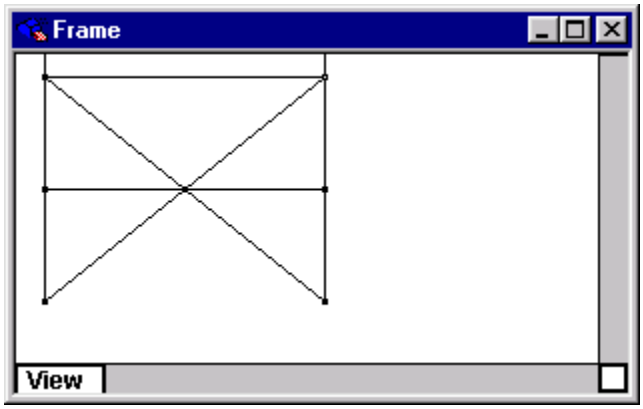
- **Choose Pan from the View Menu**

The cursor will change to a four arrowed cursor

- **Press inside the window and hold down the mouse button**



- **Drag the drawing to its new location**
- **Release the mouse button to re-draw the contents of the window.**



You can use the Ctrl-E keyboard shortcut to use the Pan command.

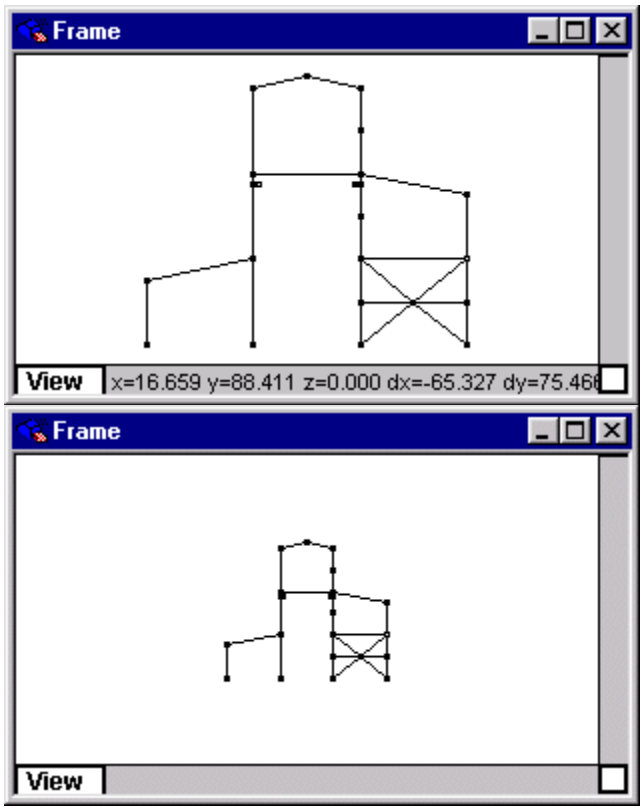
If not enough memory is available to move the image, a rectangle the size of the window will be moved around as you move the mouse.

Shrink

Shrink allows you to decrease the size of the drawing on the screen by 50%.

- **Choose Shrink from the View Menu**

The drawing will shrink down to 50% of its current size and be re-drawn.



If the mouse you are using has a wheel it can be used to dynamically zoom or shrink the view.

Size To Fit

Size To Fit automatically resizes the drawing in the front window so that the structure just fits inside the window in the current view. This is most useful after you have been zooming, panning or shrinking as it returns you to a viewing scale that just fits the frame inside the window.

Size To Fit Frame fits the whole frame inside the window.

Size To Fit Selection fits the selected members and joints inside the window.

Size To Fit Clipping fits the range of clipping inside the window.

The Home key can be used as a short cut to the Size to Fit command.

Clipping and Masking

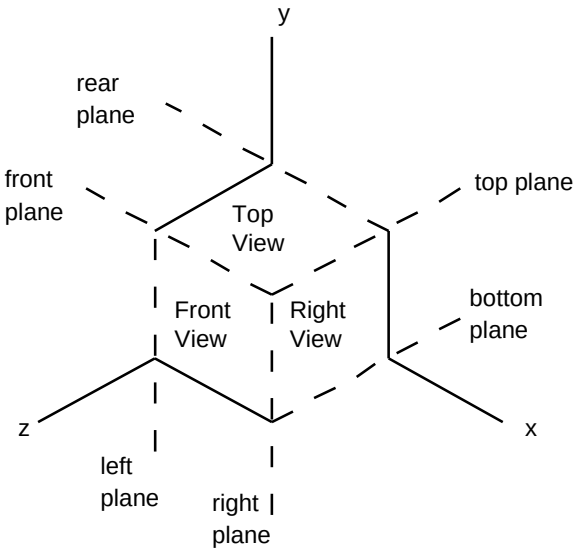
Multiframe allows you to control how much of the structure you wish to view at one time by use of two techniques named clipping and masking. Clipping allows you to define a three dimensional box which encloses the part of the structure you wish to work with while masking allows you to selectively show or hide any member or group of members in the structure.

For example, suppose you wish to view the bending moment diagram for a given column line in a frame. Without clipping, the diagram is very difficult to decipher. However, if you select the column line and then choose Clip to Selection from the Clipping sub-menu under the Edit menu, the diagram is made much clearer. Masking may be used in a similar way. When a member is grey or hidden because of clipping or masking, it cannot be selected or moved with the mouse.

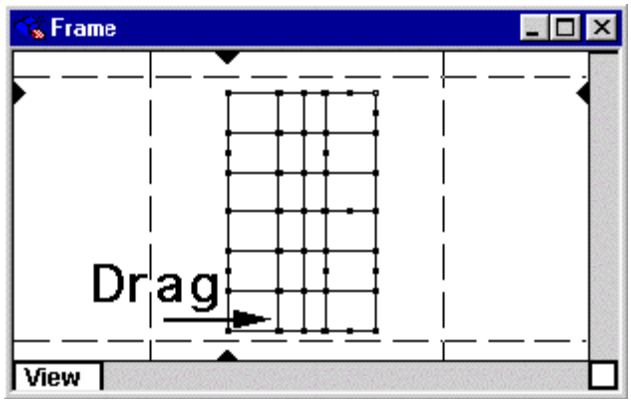
Clipping

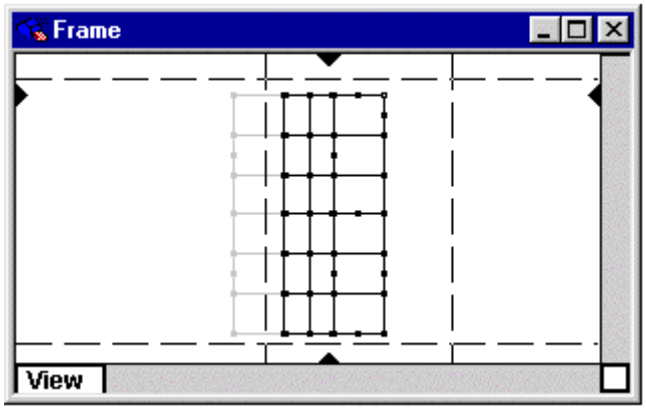
You can control the clipping of the structure in any of the two dimensional views in a window. The clipping affects the display of graphics in the Frame, Load and Plot windows. This means that you can use the clipping controls in a view of one window to make it easier to see the graphics in a different view of another window. There are two types of clipping, clipping which draws the clipped out part of the frame in grey, and clipping which makes the clipped out members completely invisible.

When clipping is turned on, the clipping bars are displayed as dotted lines in the two dimensional view. These dotted lines represent the boundaries of the clipping box.

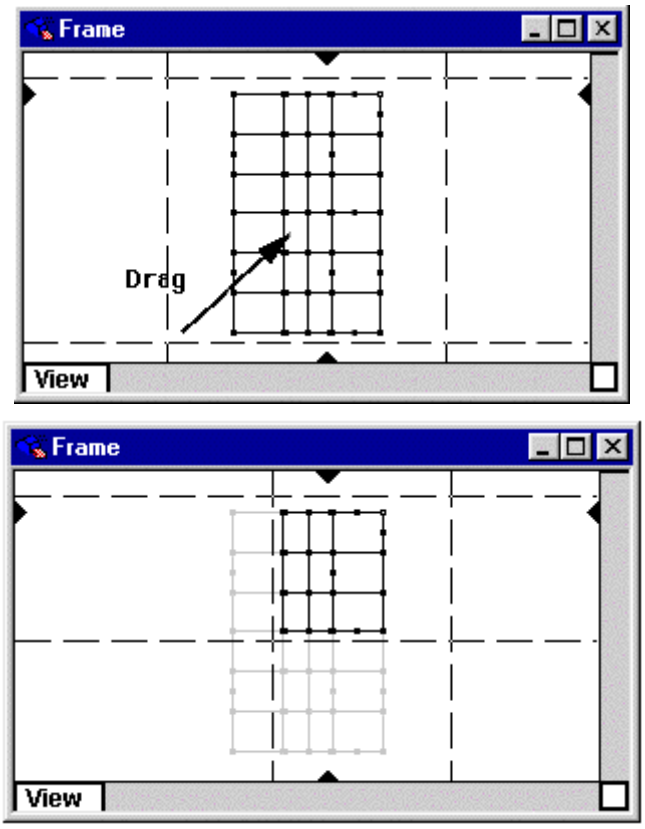


To change the boundaries of the clipping box, and therefore change which members in the structure are visible, you can press and drag on the bars with the mouse.





If you want to move two bars simultaneously, you can press and drag on the intersection between the two bars and drag them at the same time.



The small triangles located between the clipping bar on the edges of the window indicate the drawing depth. These can also be dragged using the mouse and to help align the drawing depth accurately the mouse will snap to objects and grids.

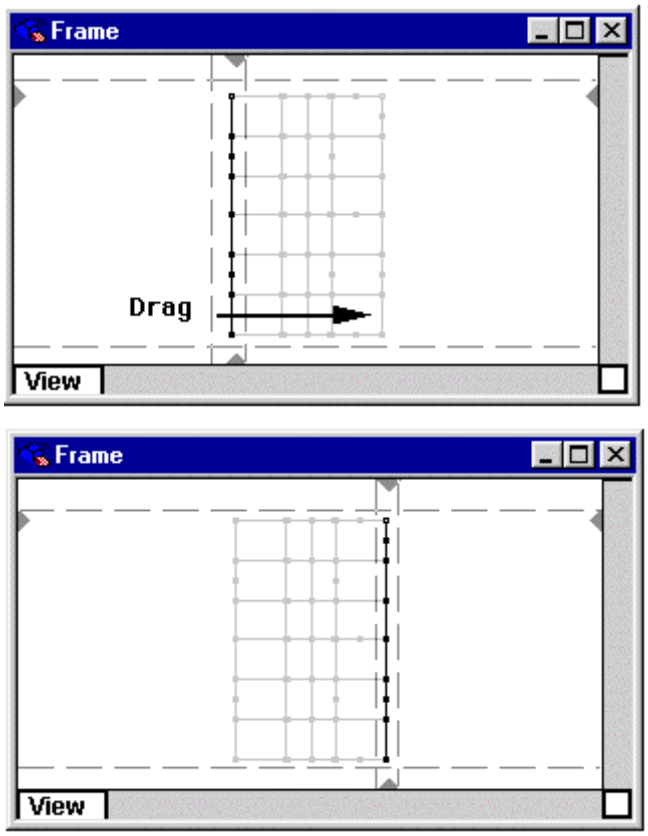
Clip To Frame

Usually you will find it convenient to start by choosing Clip To Frame from the Clipping menu. This positions the clipping bars so that they lie just outside the outer boundaries of the frame. You will then find it easy to move the appropriate bar to restrict your viewing to the part of the frame that is of interest to you.

Clip To Selection

You will also find it convenient to use the Clip To Selection command from the Clipping menu. This will position the clipping bars so that they lie just outside the farthest extents of the currently selected joints.

If you try to move a clipping bar past the position of its opposing bar (for example, move the bottom bar up past the top bar) the opposing bar will be moved to maintain a small distance between the two. This can be very useful when you want to move clipping from one floor to another or from one column line to another. You can clip on the bar farthest from the direction you wish to move and drag the two bars together.

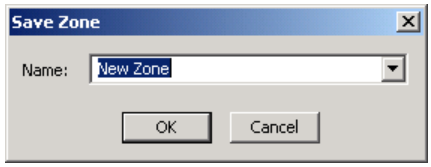


Clipping Zones

Clipping regions can be saved within Multiframe and used to restore clipping to a predefined region. This enables the user to quickly restore clipping to a particular part of the model. A clipped region is stored within Multiframe as Clipping Zone.

To save the current clipping region

- **Choose Save Clipping Zone… from the Clipping submenu in the View menu**
- **Enter a name to identify the saved region**

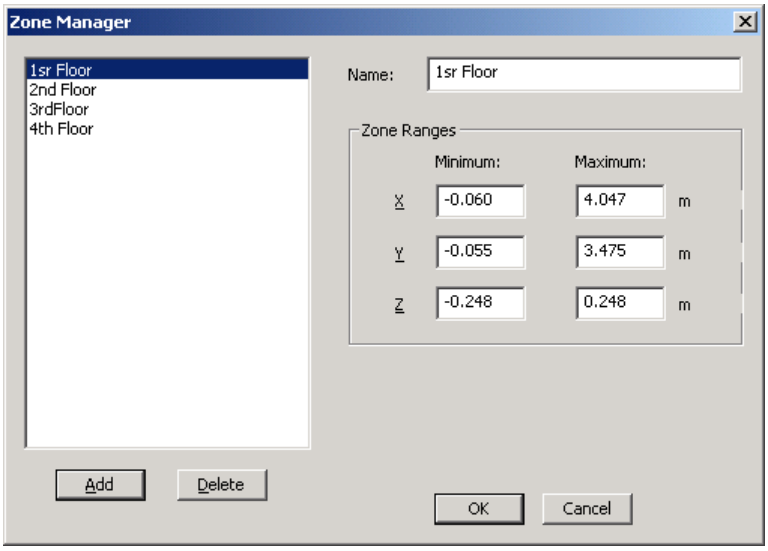


- **Click the OK button**

The current clipping regions can readily be restored to a region stored as a clipping zone.

To modify a zone

- **Choose Save Clipping Zone… from the Clipping submenu in the View menu**



Clipping zones are particularly useful when working with 3D models as separate zones can be defined for each of the 2D frames that make up the model. Each of these frames can then be quickly accessed by simply clipping to each of the zones

Masking

Masking allows you to control the visibility of the structure by selecting members and then choosing to show or hide them. If you use the Mask To Selection command in the Masking menu, this will hide all of the members in the structure except those which are selected. If you choose Mask Out Selection, the selected members will be hidden and the remaining visible members will remain visible.

Like clipping, masking affects the display of graphics in the Frame, Load and Plot windows and can also be used in a view of one window to make it easier to see the graphics in a different view of another window. Masking can also be grey or invisible however Multiframe will always ensure that clipping and masking both use the same display method. i.e. either both will display in grey or both will make hidden members invisible.

Masking is useful for situations where the area you wish to view is not rectangular in shape and therefore is not suitable for clipping.

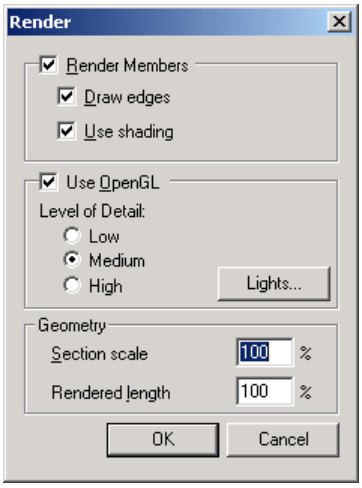
Rendering

Multiframe3D and Multiframe4D allow you to render the display of the frame in the Frame, Load and Plot windows as an aid to visualising the relative sizes and orientation of the sections in the structure. Rendering involves drawing a representation of the frame, complete with web and flange details, with hidden lines removed. Rendering can only be done in the 3D view in a window and only on the Deflection diagram in the Plot window. The number of segments drawn per member on the rendered, deflected shape is controlled by the precision set using the Plot… command from the Display menu.

To turn on rendering in the front window

- **Choose Render from the Display menu**

A dialog will appear with the rendering options.



- **Check the Render Members check box to turn on the rendering**

If you wish you can also choose whether to use shading on the sections and whether to draw separate lines for the edges of the sections.

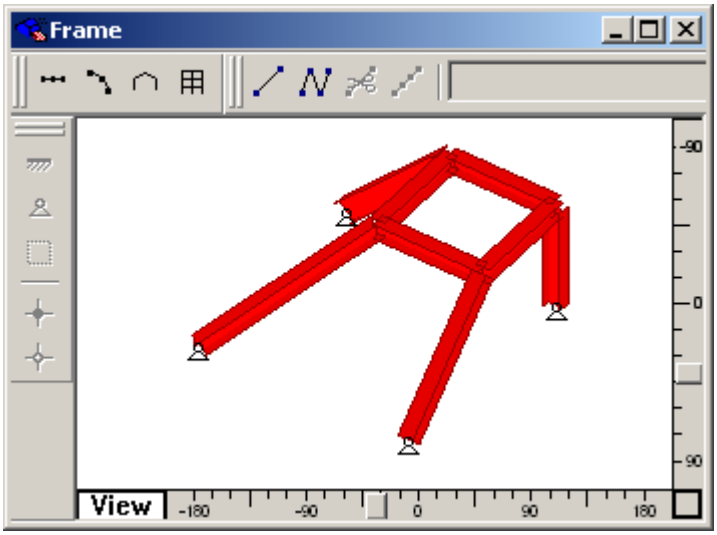
The Section scale field can be used to exaggerate the sizes of the section shapes as they are drawn in the rendered view. Because the size of sections is usually very small compared to the size of the frame, this helps make the actual shapes more clearly visible.

The Rendered length field can be used to shrink the rendered length of the members back from their actual end locations. This can help to visualise the orientation and position of members at a joint.

The OpenGL check box allows you to turn on and off the use of OpenGL rendering. This provides high speed rendering with most graphics cards and also allows use of transparency with clipping and masking. You will need to run off OpenGL rendering to print a rendered view to a printer.

- **Click the OK button**

The structure will be drawn using rendering until you turn rendering off. To turn off rendering, choose the Render command and turn off the Hidden Lines check box.



If you have clipping or masking turned on, rendering will only render the visible members. If you have drawn custom sections using Section Maker, rendering will display the actual shape of the custom section. Circular sections are displayed approximately as octagonal shapes.

Selecting Joints and Members

There are many techniques and commands that are used to selecting joint and members within a graphical window. The most common way of selecting items is by using the mouse. To select a single item

- **Click on the item**

The selected item is draw as using a thicker line size to make it appear bold. Typically this will be drawn in black but a different colour for drawing the selection can be set via the Colour command in the View menu.

To extend or reduce the selection

- **Shift-click on an unselected item to add it to the current selection**
- **Shift-click on a selected item to remove it from the current selection**

To select a group of items

- **Drag a rectangle which encloses the items to be selected**

If the rectangle is defined by dragging the cursor from the left to the right, a bounding selection will be performed and only the items within the rectangle will be selected. If shift is held down while dragging, the existing selection will be extended with the additional unselected items that are contained within or intersect the rectangle.

When performing a drag selection there are two techniques for items. The first is by using a rectangular box as described above. An alternative to this is to use a line selection in which the user drags the end of a straight line across the screen. In this case all members that intersect this line are selected and well as all joints connected to these members.

The selection tool to be used when performing a drag selection is picked using the View Toolbar. Clicking on the dotted square or dotted line buttons chooses the corresponding selection tool.

The Select menu contains a number of commands for helping to select items in the model. Member and Joint may be selected by specifying their number or by a label.

Selecting Load Panels

There are many techniques and commands that are used to selecting load panels within a graphical window. The most common way of selecting items is by using the mouse. To select a single item

- **Click on the item**

The selected panel is draw with dark pink colour to make it appear highlighted, but a different colour for drawing the selection can be set via the Colour command in the View menu.

To extend or reduce the selection

- **Shift-click on an unselected panel to add it to the current selection**
- **Shift-click on a selected panel to remove it from the current selection**
- **Shift-drag to invert the selection in the selection rectangle**

To select a group of items

- **Drag from left to right a rectangle which encloses the panels to be selected**
- **Drag from right to left to select all load panels which intersect or are contained inside the selection rectangle**

If the rectangle is defined by dragging the cursor from the left to the right, a bounding selection will be performed and only the items within the rectangle will be selected. If shift is held down while dragging, the existing selection will be extended with the additional unselected items that are contained within or intersect the rectangle.

When performing a drag selection there are two techniques for items. The first is by using a rectangular box as described above. An alternative to this is to use a line selection in which the user drags the end of a straight line across the screen. In this case all members that intersect this line are selected and well as all joints connected to these members.

Selecting Patches

There are several techniques and commands that are used to select patches within a graphical window. The most common way of selecting items is by using the mouse. To select a single item

- **Click on the item**

The selected patch is drawn in a darker colour . A different colour for depicting the selection can be set via the Colour command in the View menu.

To extend or reduce the selection

- **Shift-click on an unselected patch to add it to the current selection**
- **Shift-click on a selected patch to remove it from the current selection**
- **Shift-drag to invert the selection in the selection rectangle**

To select a group of items

- **Drag from left to right a rectangle which encloses the patches to be selected**
- **Drag from right to left to select all patches which intersect or are contained inside the selection rectangle**

If the rectangle is defined by dragging the cursor from the left to the right a bounding selection will be performed. Only the items within the rectangle will be selected.

If the rectangle is defined by dragging the cursor from right to left an intersect selection will be performed. Any item that intersects the rectangle will be selected.

If shift is held down while dragging, the existing selection will be extended with the additional unselected items.

A second technique for performing a drag selection is to use a line selection. When the line selection tool is chosen the user drags the end of a straight line across the screen. All patches that intersect this line and all joints connected to these patches are selected.

Selecting Plates

There are several techniques and commands that are used to select plates within a graphical window. The most common way of selecting items is by using the mouse. To select a single item

- **Click on the item**

The selected plate is drawn in a darker colour . A different colour for depicting the selection can be set via the Colour command in the View menu.

To extend or reduce the selection

- **Shift-click on an unselected plate to add it to the current selection**
- **Shift-click on a selected plate to remove it from the current selection**
- **Shift-drag to invert the selection in the selection rectangle**

To select a group of items

- **Drag from left to right a rectangle which encloses the plates to be selected**
- **Drag from right to left to select all plates which intersect or are contained inside the selection rectangle**

If the rectangle is defined by dragging the cursor from the left to the right a bounding selection will be performed. Only the items within the rectangle will be selected.

If the rectangle is defined by dragging the cursor from right to left an intersect selection will be performed. Any item that intersects the rectangle will be selected.

If shift is held down while dragging, the existing selection will be extended with the additional unselected items.

A second technique for performing a drag selection is to use a line selection. When the line selection tool is chosen the user drags the end of a straight line across the screen. All plates that intersect this line and all joints connected to these plates are selected.

Selecting Patch Openings

There are several techniques and commands that are used to select openings within a graphical window. The most common way of selecting items is by using the mouse. To select a single item

- **Click on the item**

The selected opening is drawn in a darker colour. A different colour for depicting the selection can be set via the Colour command in the View menu.

To extend or reduce the selection

- **Shift-click on an unselected opening to add it to the current selection**
- **Shift-click on a selected opening to remove it from the current selection**
- **Shift-drag to invert the selection in the selection rectangle**

To select a group of items

- **Drag from left to right a rectangle which encloses the openings to be selected**
- **Drag from right to left to select all openings which intersect or are contained inside the selection rectangle**

If the rectangle is defined by dragging the cursor from the left to the right a bounding selection will be performed. Only the items within the rectangle will be selected.

If the rectangle is defined by dragging the cursor from right to left an intersect selection will be performed. Any item that intersects the rectangle will be selected.

If shift is held down while dragging, the existing selection will be extended with the additional unselected items.

A second technique for performing a drag selection is to use a line selection. When the line selection tool is chosen the user drags the end of a straight line across the screen. All openings that intersect this line and all joints connected to these openings are selected.

Select Patch Edges

There are many techniques and commands that are used to selecting patch edges like selecting a patch edge within a graphical window. The most common way of selecting items is by using the mouse. To select a single patch edge

- **Click the area inside the patch edge**

The selected patch edge is drawn with the current selection colour to make it appear highlighted. A different colour for drawing the selection can be set via the Colour command in the View menu.

To extend or reduce the selection

- **Shift-click on an unselected patch edge to add it to the current selection**
- **Shift-click on a selected patch edge to remove it from the current selection**
- **Shift-drag to invert the selection in the selection rectangle**

To select a group of items

- **Drag from left to right a rectangle which encloses the patch edges to be selected**
- **Drag from right to left to select all patch edges which intersect or are contained inside the selection rectangle**

When performing a drag selection there are two techniques for items. The first is by using a rectangular box as described above. An alternative to this is to use a line selection in which the user drags the end of a straight line across the screen. In this case all patch edges that intersect this line are selected.

Select Opening Edges

There are many techniques and commands that are used to selecting opening edges like selecting a opening edge within a graphical window. The most common way of selecting items is by using the mouse. To select a single opening edge

- **Click the area inside the opening edge**

The selected opening edge is drawn with the current selection colour to make it appear highlighted. A different colour for drawing the selection can be set via the Colour command in the View menu.

To extend or reduce the selection

- **Shift-click on an unselected opening edge to add it to the current selection**
- **Shift-click on a selected opening edge to remove it from the current selection**
- **Shift-drag to invert the selection in the selection rectangle**

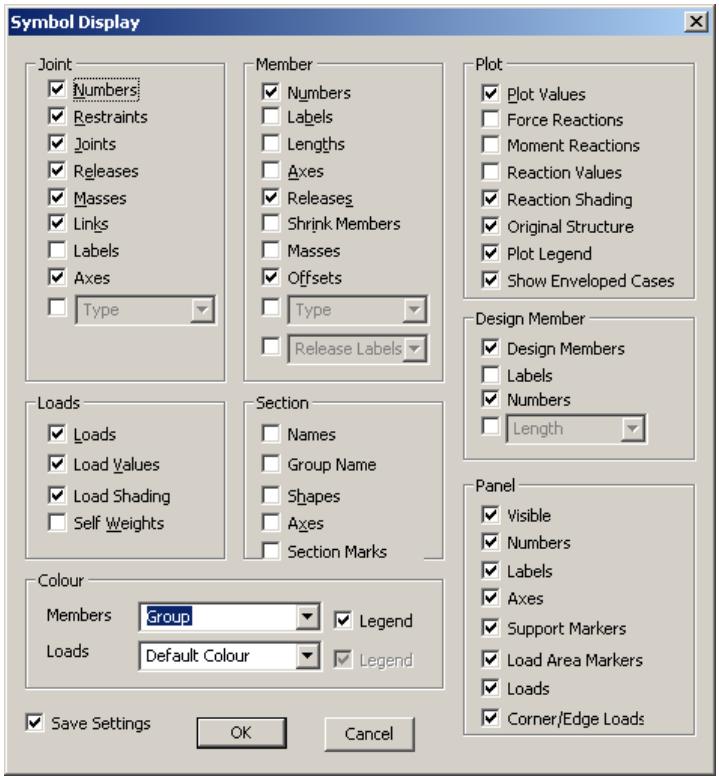
To select a group of items

- **Drag from left to right a rectangle which encloses the opening edges to be selected**
- **Drag from right to left to select all opening edges which intersect or are contained inside the selection rectangle**

When performing a drag selection there are two techniques for items. The first is by using a rectangular box as described above. An alternative to this is to use a line selection in which the user drags the end of a straight line across the screen. In this case all opening edges that intersect this line are selected.

Symbols

The information displayed in the graphical windows may be modified by the user via the Symbols Dialog. This dialog is accessed via the Symbols command in the Display menu.



This dialog has options to display information related to the joints, members and panels in the model. For joints, this includes the properties of the joint such as its number, label or local axes. Other symbols can be displayed to indicate other information related to joints such as restraints, joint masses, and linked joints. Similar options are available for members and panels; they may be displayed with their number, label, length and local axes. They may also be annotated with their section names or shapes. Symbols representing other properties that are related to the ends of the members may also be shown an include end releases and offsets.

Other setting available via this dialog allow the display in the plot and loads windows to be customised. A number of options are available that specifies how and if loads are display and whether the loads are labelled with their magnitudes. Similar options are available for controlling the display of reactions and labels in the Plot window.

The colour used to draw each element in the Frame window is also set via the Symbols dialog. Members in the Frame window can be displayed using a number of colouring schemes that use different colours to distinguish members based upon their section, section shape or label. Grouping of members may also be highlighted by displaying members using the colour of the group in which they are contained. A legend summarising the colours used in drawing the model may also be displayed.

The symbols toolbar provides a number of buttons to toggle the display of the more frequently used symbols.

Legends

Legends displaying the meaning of colours used to display members and loads can be displayed by selecting the corresponding option in the Symbol Display dialog available from the Display menu.

A feature of some of the legends used in Multiframe is that you can use the legend to modify the selection in the current window. Double clicking on the text of an item in the legend will select the corresponding items in the model. For example, this can be useful for selecting all the members in a group, all the members with the same label or all he members with the same section type.

Member Legend

Members in the Frame and Plot window can be displayed in colour. The colour of the member is selected via the Symbol Display dialog (available from the Display menu) by choosing a colour scheme. The following options are available:

Default colour

When this option is selected, the default colour settings will be used. The default colour settings can be set via the View | Colour dialog.

Section

When this option is selected, the members' colour will be displayed as per the colour of the Section type. The legend will list all different section types used in the structure.

Section Group

When this option is selected, the members' colour will be displayed as per the colour of the group their assigned section type belongs to. The legend will list all different section groups that the members used in the structure belong to. Section groups are for example UB, Channels, Equal Angles etcetera.

Section Shape

When this option is selected, the members' colour will be displayed as per the colour of the shape. The legend will list all different shapes of the members used in the structure. Shapes are for example Tube, I section, Channel, etcetera.

Label

When this option is selected and members have been labelled, the members' colour will be displayed as per the label colour. The legend will list all different labels.

Type

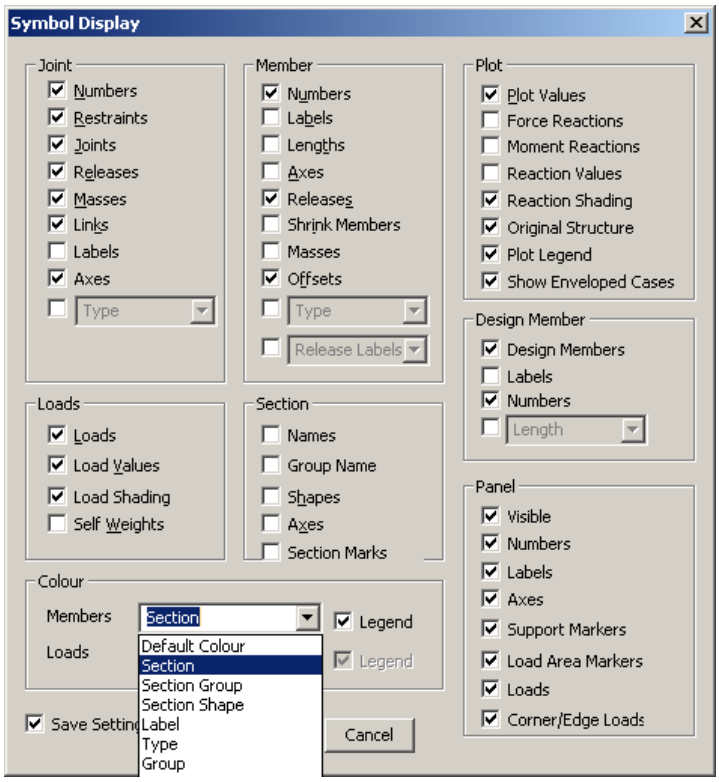
When this option is selected, the members' colour will be displayed as per the member type colour. The legend will list the different types of members used in the structure. Types are for example "Tension only" or "Compression only".

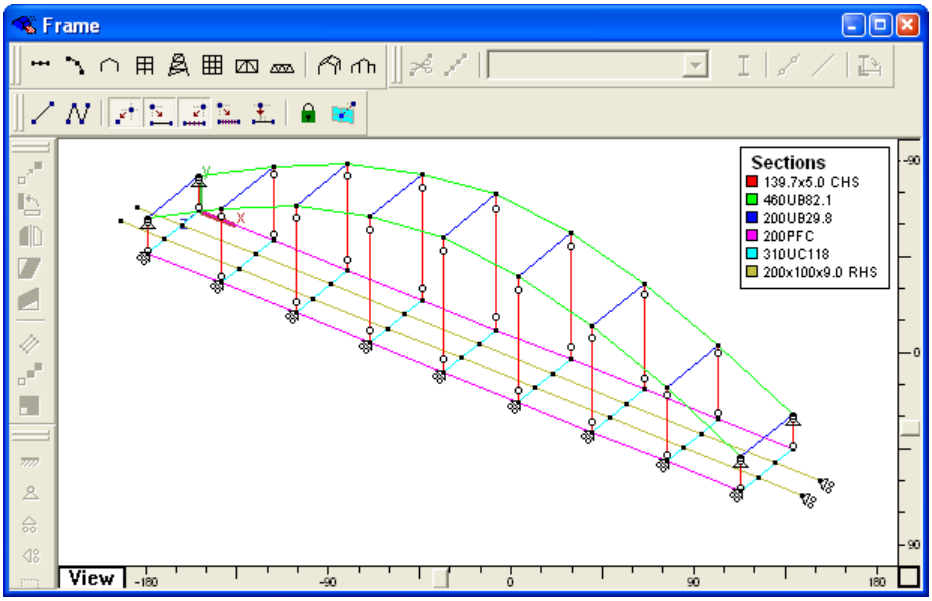
Group (in current group set)

When this option is selected and members have been assigned to groups, members' colour will be displayed as per the group they belong to. The legend will list all different groups used in the current group set. For more information on Groups and Group Sets, please see Grouping on page 107.

Component

When this option is selected and members have been assigned to components, the members' colour will be displayed as per the component's colour. The legend will list all different components.





Tip

You can use the legend to quickly select members that share the same property or label. For example: if you want to select all members with a Tube shape, you set the Colour scheme to Shape in the Display | Symbols dialog and then double click on “Tube” in the legend.

Load Legend

The colour used to display each load in the Load Window is set by selecting the colour scheme in the Symbol Display dialog (available from the Display menu). As with members, various properties of the loads can be displayed graphically. The following options are available:

Load type

Display the colour of the load by the Load type. Load types are for example: Local Member Point Loads, Local Joint Loads etcetera.

Direction

Display the colour of the loads by the direction the load is in.

Localised index

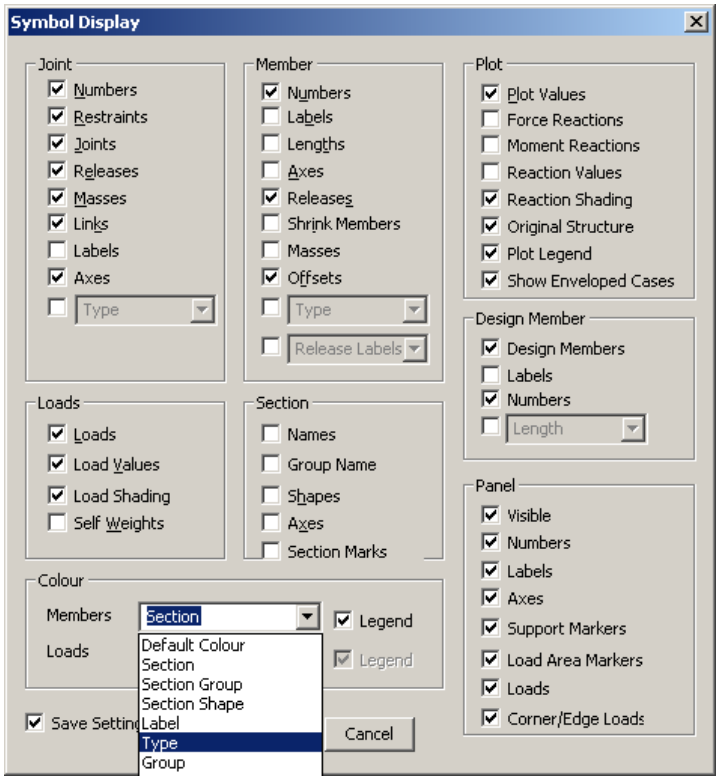
When several loads are applied on one member, their display can be set so that the 1st, 2nd, 3rd, … loads on each member are shown in different colours.

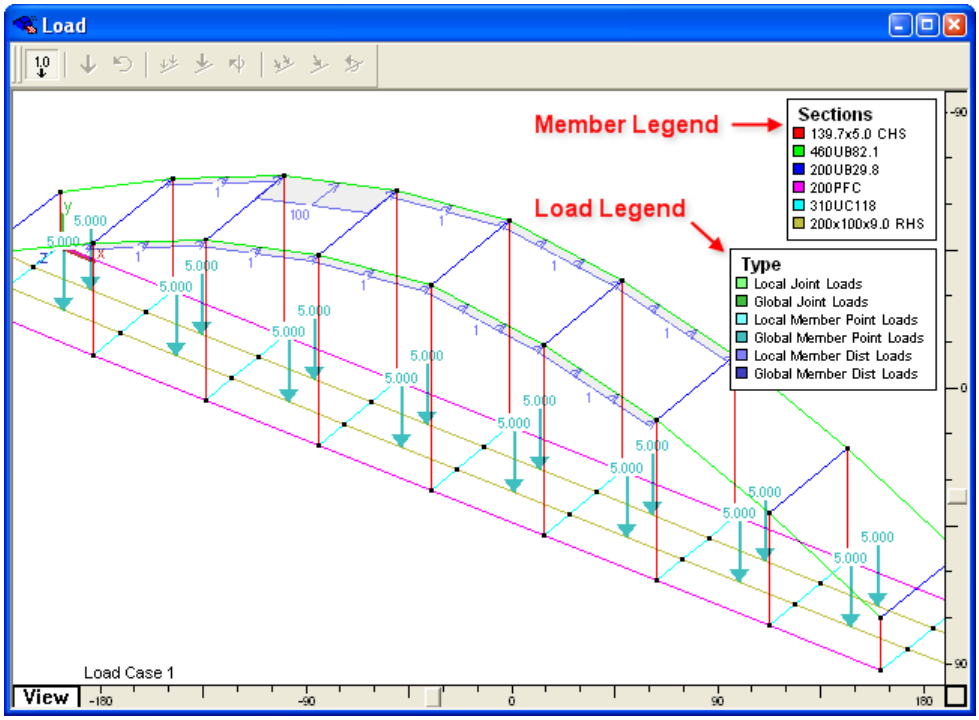
Loadcase

Display the colour of the loads by base load case from which load originates.

Loading type

Display the colour of the loads by the Loading type. Loading types can be for example dead, live, etc





Note

The Load legend is activated by selecting the check box next to the load colour scheme in the Symbol Display dialog (available from the Display menu).

Legend Properties

Legends are drawn in the corners of a window, by default this is the top right hand corner. The position of the legends can be modified by the user via the Legend submenu contained within the Display menu. The font size and style used to draw the legend can also be customised using the commands in the Legend submenu.

In many instances it is also possible to customise the colour of each item in the legend. This can be done via the submenu or by double clicking on the colour box in the legend. Note that many legends use the same list of legend colours and that editing the colour of an item in one legend will be affect the colour in many legends.

A popup menu can also be activated by right clicking the mouse on the legend. The popup menu has options to edit the colours, style and position of the legend and to choose the current colour scheme.

Centre of Gravity

Multiframe calculates and displays the centre of gravity of the model while you work. The CG can be displayed graphically in the drawing windows and is also displayed at the bottom of the Sections table in the Data window.

As of Multiframe v14, the Centre of Gravity calculation now includes Joint Masses.

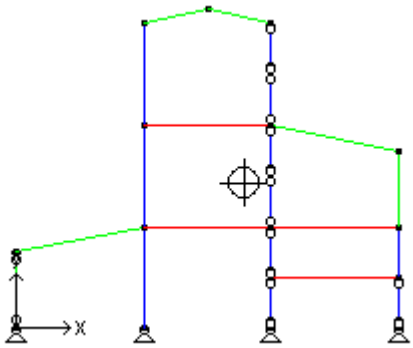
DXF Background

It is now possible to load a DXF file and display it in the background behind your Multiframe model. The command can be accessed from the Import sub-menu under the File menu. A DXF Background menu item in the View menu and a corresponding icon at the end of the View toolbar can be used to turn the display of the DXF background image on and off.

You can control the colour of the DXF background image using the View-Colour command and selecting the item at the end of the list.

Before importing a DXF file, you should be aware of some pre-processing that you should do before loading into Multiframe.

- Only lines, linear polylines and arcs will be loaded
- All blocks should be exploded prior to import
- All polylines containing arcs should be exploded
- Units in the DXF file should match units in the Import dialog which appears in Multiframe when loading the file



Section Library - C:\Documents and Settings\phil\Desktop\SectionsLibrary.slb

	Section	Group	Length m	Number Us	Total Length m	Mass/L kg/m	Total Mass kg
15	125x75x10	Unequal An	7.214	4	28.855	14.200	409.744
16	250UC72.9	UC	5.657	2	11.314	72.900	824.769
17	**Total**				**1713.303**		**90080.922**
18	**CGx**		**17.828**				
19	**CGy**		**11.408**				
20	**CGz**		**23.709**				

Spring Members | Joint Masses | Sections | Materials | Step Load

A CG icon shortcut in the View toolbar can be used to turn the CG icon on and off.

The centre of gravity calculations include the mass of all of the members in the model as well as the mass of any plate elements in the model and joint masses.

Creating a Structure

Creating a structure with Multiframe involves defining the geometry, restraints, section types and the connections of the structure by drawing in the Frame window. You can also use the pre-defined generation aids to automatically construct portal frames or geometrically regular structures.

This section begins with a description of the drawing techniques you can use to create a structure. This is followed by a summary of how to select joints and members in the structure and how to use Multiframe capabilities to assist in generating commonly used structures. A description of the commands that can be used to specify data defining restraints, section properties and members with pins or moment releases, completes the section.

Drawing

Members of the structure may be drawn directly by using the mouse to define the beginning and end points of a member. The scale at which this drawing is carried out may be specified by choosing the Size… command from the View Menu. The maximum and minimum coordinates to be used in the x, y and z directions may be entered and Multiframe will scale these coordinates to the current size of the Frame window. All movements in the window are accompanied by a display of the current pointer coordinates in the lower left hand corner of the window. All coordinates are shown in the current length units as set using the Units command from the View menu.

Similar to members' drawing, load panels can also be drawn directly by using panel drawing commands from the Geometry Menu or shortcuts. The panels can be either triangle or quadrilateral, but have to be planar.

Patches can also be drawn directly by using patch drawing commands from the Geometry Menu or shortcuts. The patches can be either triangle or quadrilateral, but have to be planar.

Openings can also be drawn directly by using opening drawing commands from the Geometry Menu or shortcuts. The openings can be N-sided, but have to be planar and are associated with a patch.

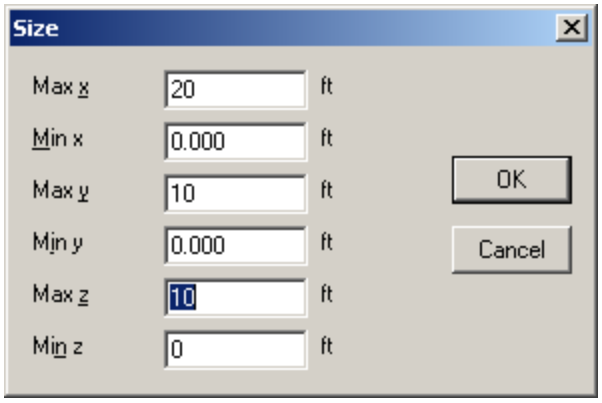
Setting the Size

Before starting drawing, you will need to set up the drawing area for the size of frame you intend working with. To do this

- **Choose Size… from the View Menu**

A dialog box will appear with the dimensions of the structure.

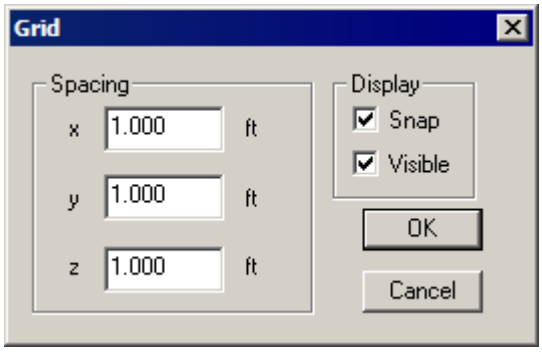
- **Enter the maximum and minimum coordinates you wish to use in each direction**
- **Click the OK button**



Drawing Grid

Multiframe has a built-in facility to allow you to have your drawing automatically align with an evenly spaced grid. You can control the spacing, display and use of this grid with the Grid… command from the View Menu.

- **Choose Grid… from the View Menu**

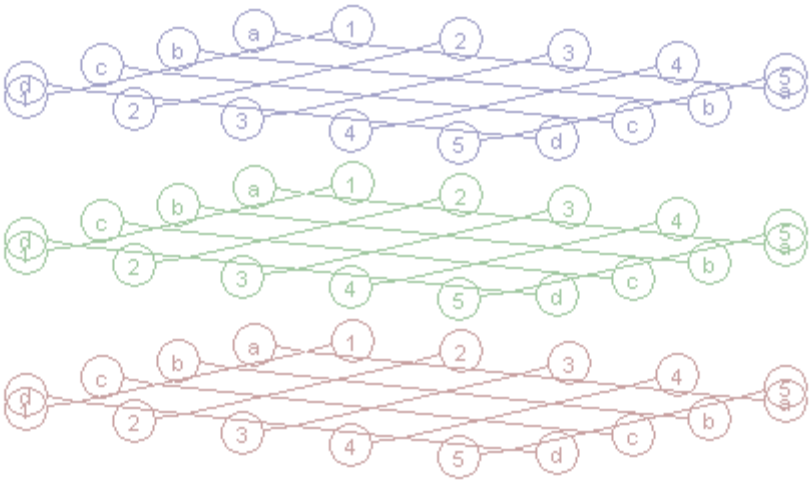


- **Enter values for the x, y and z spacing of the grid**
- **Click on the Visible checkbox if you want the grid to be displayed in the Frame Window.**
- **Click on the Snap checkbox if you want drawing to align to the grid.**
- **Click the OK button**

If you want to have the joints automatically align with the grid, but do not want to see the grid, simply unselect the Visible button before clicking OK. Similarly you can select the Visible to have the grid displayed as a visual guide, but unselect the Snap to disable the automatic alignment with the grid.

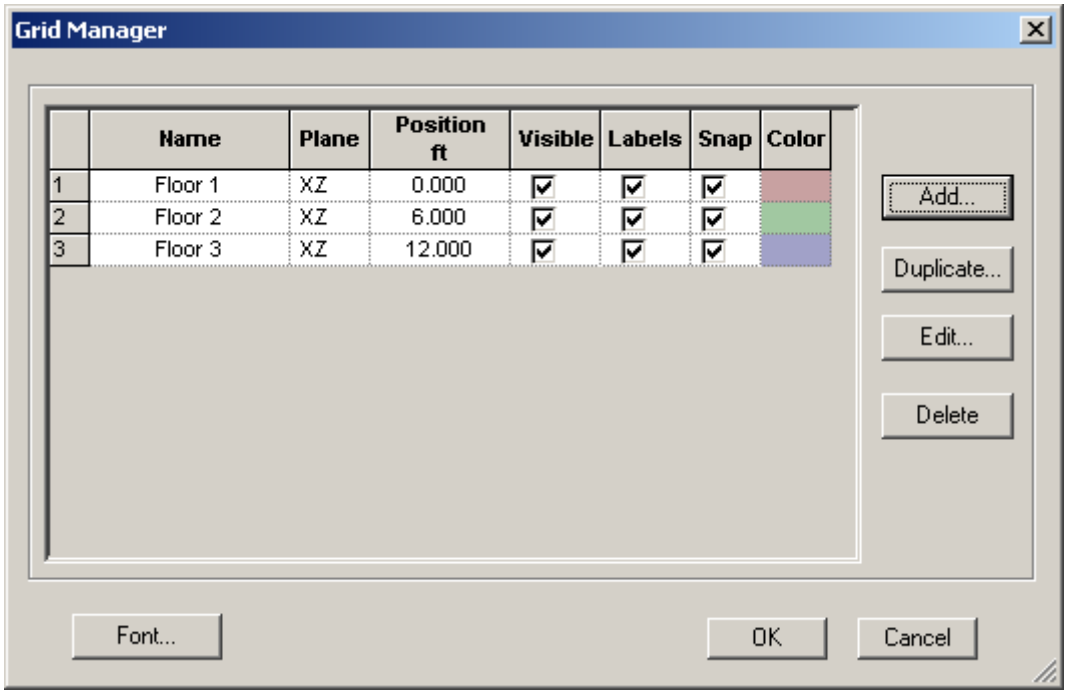
Structural Grid

Multiframe also has a facility to define a 3D structural grid that allows you to have your drawing automatically align with a predefined grid associated with the building.



The structural grid is composed of a number of planar grids that nominally represent a floor within your building. You can control the size, location, labelling, and use of these grids with the Structural Gridcommand in the View Menu.

- **Choose Structural Grid… from the View Menu**



The Grid Manager dialog appears that allows you to add, edit, delete and modify the behaviour of grids.

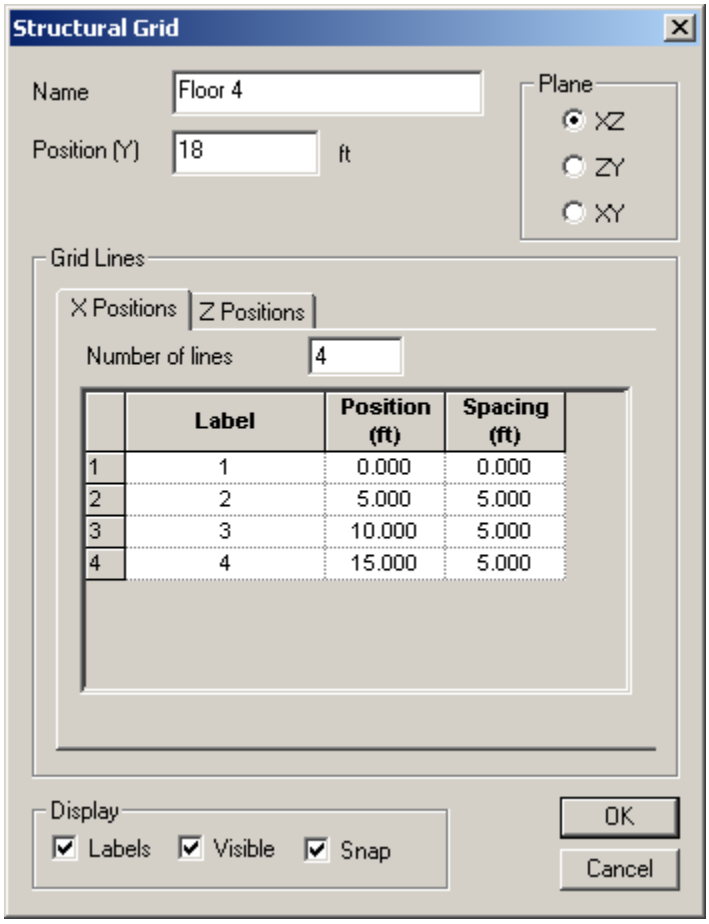
The table of grids can be used to directly edit some properties if the individual grids that make up the structural grid. The behaviour and display of each grid can be controlled individually. As with the drawing grid, if you want to have the joints automatically align to a grid, but do not want to see the grid, simply unselect the Visible button before clicking OK. Similarly you can select the Visible to have the grid displayed as a visual guide, but unselect the Snap to disable the automatic alignment with the grid.

Adding a Grid

To add a new grid

- **Click on the Add… button**

The Structural Grid dialog is displayed which is used to define the properties of the new grid.



- **Enter the name of the grid**
- **Choose orientation of the grid by selecting a plane.**
- **Specify the position of the grid by setting the location of the grid along the axis perpendicular to the plane of the grid.**
- **For each of the two directions in the plane of the grid specify the number of grid line and there location.**
- **Click the OK button**

Editing a Grid

To edit an existing grid

- **Select the row in the table of grids corresponding to the grid to be edited. Alternatively just a cell in this row can be selected.**
- **Click on the Edit… button**

The Structural Grid dialog is displayed which is used to modify the properties of the grid (see above).

Deleting a Grid

To delete an existing grid

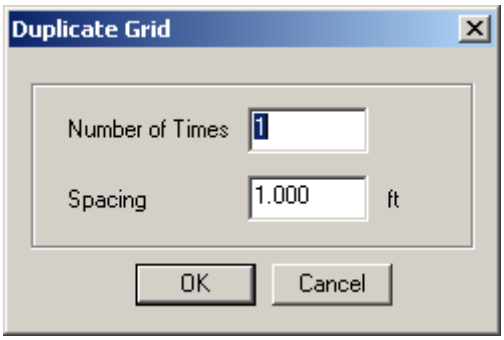
- **Select the rows in the table of grids corresponding to the grids to be deleted.**
- **Click on the Delete… button**

You will be prompted to confirm that you wish to delete the selected grids.

Duplicating a Grid

To edit an existing grid

- **Select the rows in the table of grids corresponding to the grids to be duplicated.**
- **Click on the Duplicate… button**



- **Enter the number of times to duplicate the grids**
- **Enter the distance to increment the position of the duplicated grids.**
- **Click the OK button**

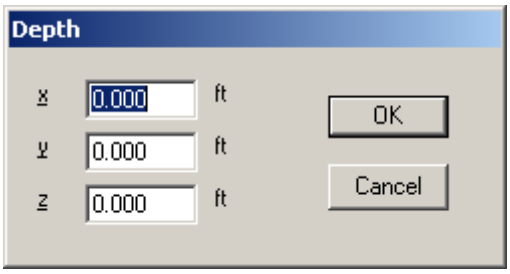
Drawing Depth

When you draw in a two dimensional view it is necessary to specify the depth at which you are drawing. For example, when drawing in the Top view, the y coordinate must be specified for any members drawn.

To change the current drawing depth

- **Choose Drawing Depth… from the View menu**

A dialog box will appear with the current depth selected.



Type in a new value for the depth and click OK to set the new depth.

Snapping to Joints and Members

Drawing in Multiframe will also automatically align to joints and members in your model.

You can control which parts of the model the cursor will snap to when drawing via the Drawing Settings dialog from the Geometry menu. This dialog gives you options to snap to

- **Joints**
- **Members**
- **Member quarter points**
- **A customised number of points along members**
- **Perpendicular point on member**

The option to snap to a perpendicular point on a member is only used to snap the second end of a line been drawn such that the line is drawn perpendicular to the existing member.

When drawing in a 3D view the drawing defaults to be located in the current working plane. This plane is the plane oriented most perpendicular to the current viewing direction which passes through the current drawing depth. The axes indicating the origin of the coordinate system highlight the axes that identify the orientation of the drawing plane. When the cursor snaps to an existing item in the model the drawing will snap to this point in 3D space.

When drawing is a 2D view snapping to existing pasts of the model out of the current drawing plane can be a problem. For example, when trying to draw a planar structure in front of an existing part of the model the cursor may snap to existing objects which can move the drawing from the current working plane. To avoid this problem all drawing can be forced to occur in the current working plane by selecting the 'Constrain drawing to working plane option' in the Drawing Settings dialog. When this option is selected, the cursor will still snap to existing parts of the model but drawing will be projected back to the current drawing depth.

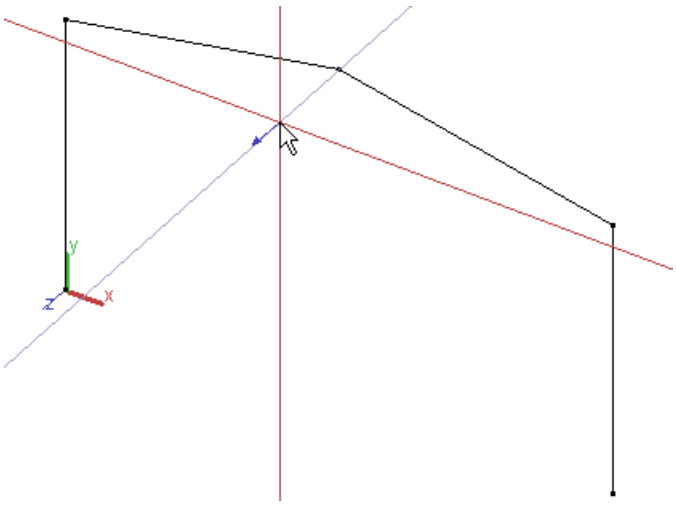
When drawing or dragging you can stop the cursor snapping to joints, members and grid points by simply holding down the ALT key while performing the operation.

Dynamic Line Constraints

By default, drawing will occur on the current working plane which is the plane aligned with the global axes, that is most normal to the current viewing direction and passing through the current drawing depth. To help with the alignment of the model and to allow drawing out of the current working plane Multiframe uses dynamic line constraints. When drawing a member, if the cursor has snapped to a joint and the mouse moved in the direction of one of the global axis, drawing will be constrained to occur along this direction in 3D space.

The constraint of drawing to the global axis direction described above is in fact part of a more general concept for automatically constraining the direction the mouse moves in 3D space. The drawing direction can also be constrained to occur in the direction of members connected to a joint. If the mouse is moved in the direction of a member attached to the joint then drawing will be constrained to the direction of that member in 3D space.

When movement is constrained the direction of a global axis or element, a small arrow will be drawn at the cursor pointing in the direction of the line of the constraint. The colour of the arrow indicates is the constraint is in the direction of a global axis or a member.

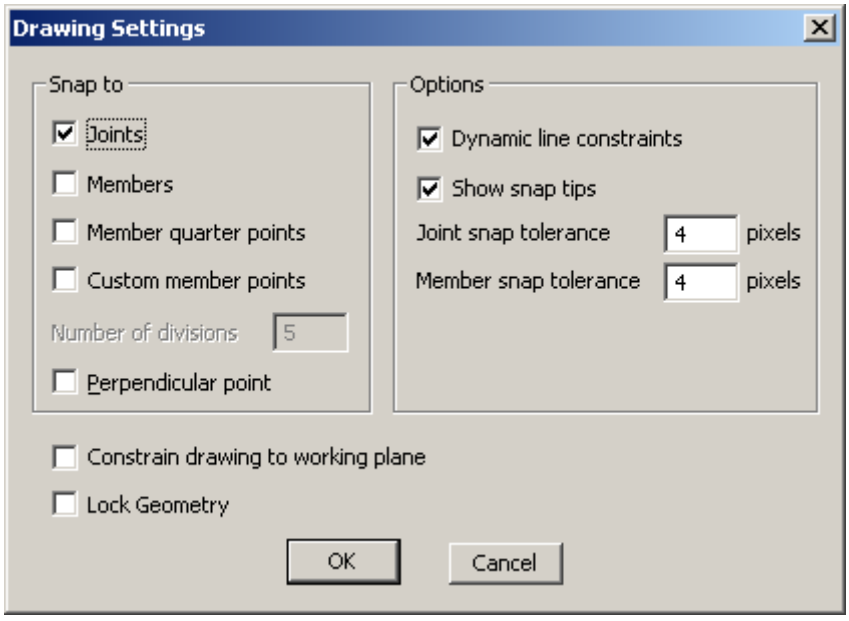


The drawing constraint will be deactivated if the cursor is moved to far away from the direction of the constraint. This can be avoided by holding down the shift key and this will force the current restraint to remain active.

Dynamic line constrains can be disabled via the new Drawing Settings dialog.

Drawing Settings

The Drawing Setting dialog from the Geometry menu allows you to modify options associated with snapping to objects in the model, the use of dynamic line constraints and other options associated with graphically interacting with the model.



The Lock Geometry option in this dialog provides a means for stopping the geometry and topology of the frame from being modified. This includes disabling all drawing, dragging and generation of structural members.

This dialog also provides some options for controlling how far the cursor will move to snap to an object in the model. The Joint snap tolerance specifies how close the cursor has to be to a joint to snap to the joint. Similarly the Member snap tolerance specifies how close the cursor has to me to a member (or point on a member) to snap to the member. In general the member snap tolerance should be equal to or smaller than the joint snap tolerance. By default these values are quite small but for most models tolerances of 20 pixels or greater can be used.

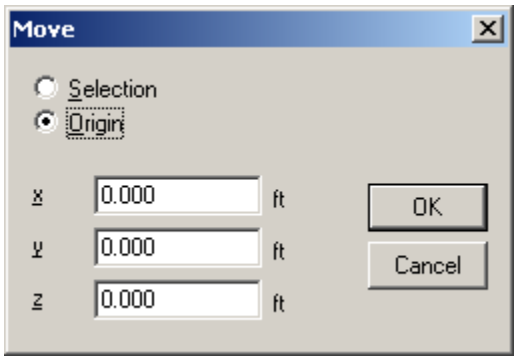
Setting the Origin

Multiframe has an origin for its coordinate system indicated by a symbol of x, y and z axes drawn in the Frame window. You can control the display of these axes by using the Axes command from the View Menu.

You can also change the location of the origin, relative to a structure.

- **Choose Move …from the Geometry menu**

A dialog box will appear allowing you to enter x, y and z coordinates for the new position of the origin.

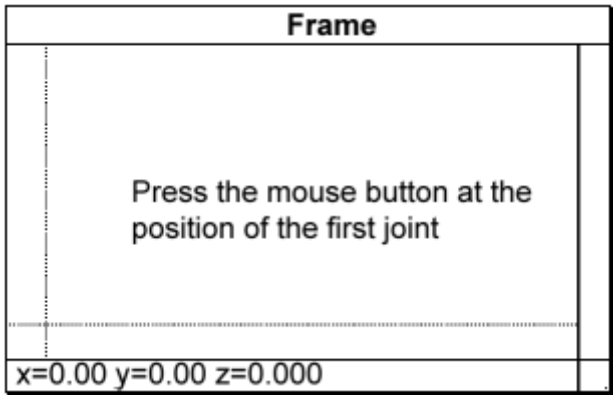


This command is useful for creating structures that are made up of larger sub-structures. You can shift the origin to a convenient location prior to doing the drawing for each sub-structure.

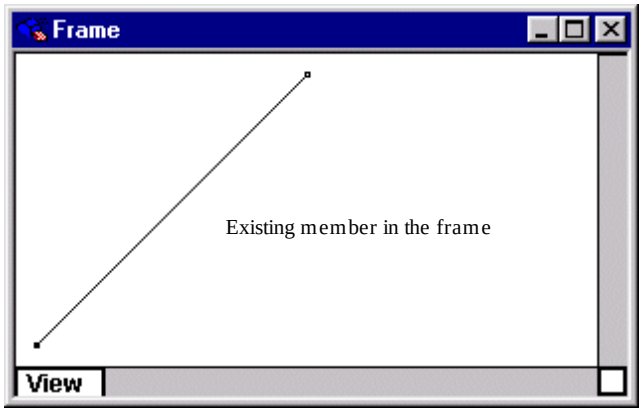
Adding a Member

To add a member to the structure

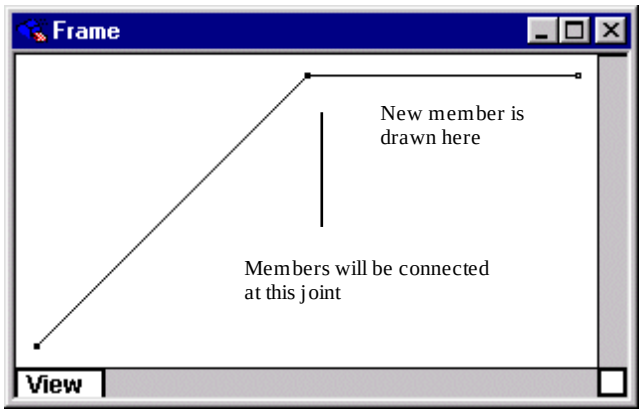
- **Choose Add Member from the Geometry menu**
- **Or Click on the Add member icon in the Member toolbar**
- **Move the pointer to the position of the first joint**



- **Press the mouse button and drag to the position of the second joint (Windows users have the option of just clicking on the first joint instead of dragging)**

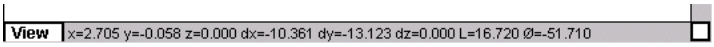


- **Release the mouse button (You also have the option of just clicking on the second joint to complete the new member)**



Holding down the shift key while drawing a member constrains the new member to be either vertical or horizontal or at 45 degrees.

When you draw the new member, the coordinates of the mouse will be displayed in the bottom left corner of the window. Also, the distance and slope of the current position of the mouse from the last point will be displayed.



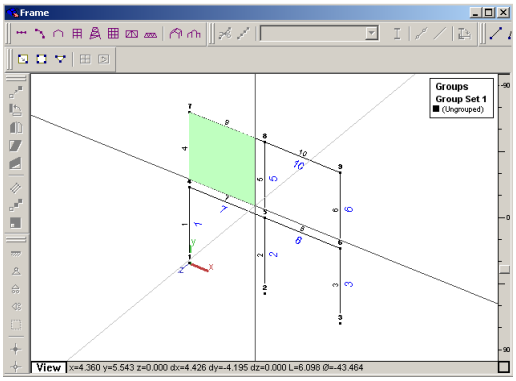
When drawing in the 3D view, you can only draw a member between two existing joints.

If the grid is turned on while you draw the member, the position of the joints will snap to the grid. The depth of the member will depend on the setting of the depth for the current view. See Depth above for an explanation of how to set this depth.

Adding a Load Panel

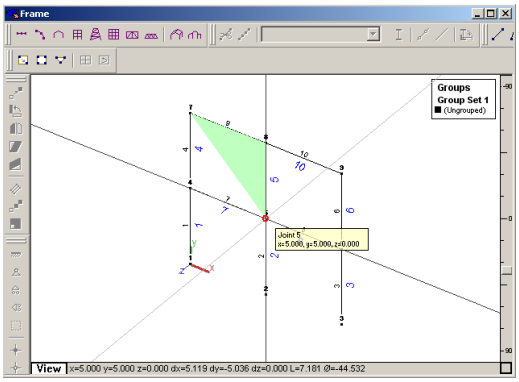
There are several ways for users to add a load panel to the structure. From Geometry Menu, a user can:

- **Choose Add rectangle Load Panel from the Geometry menu**
- **Click on two nodes at diagonally opposite corners of the panel.**



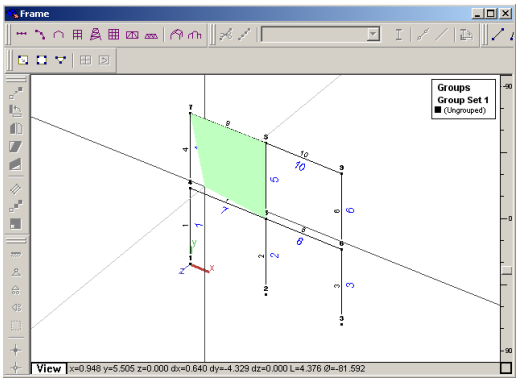
To add a triangular load panel

- **Choose Add 3-node Load Panel from the Geometry**
- **Click in anti-clockwise order the three nodes that define the corners of the load panel**



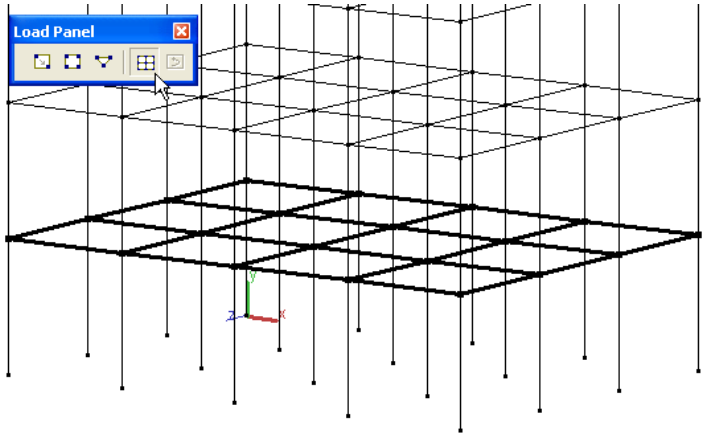
To add a quadrilateral load panel

- **Choose Add 4-node Load Panel from the Geometry menu**
- **Click in anti-clockwise order the four nodes that define the corners of the panel.**

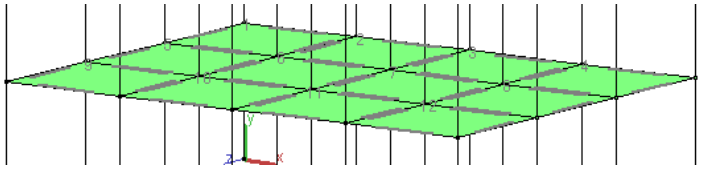


To automatically add load panels between all of the members in a planar region (eg a floor or frame)

- **Select the members which define the planar region**



- **Choose Auto-generate Load Panels from the Geometry menu**

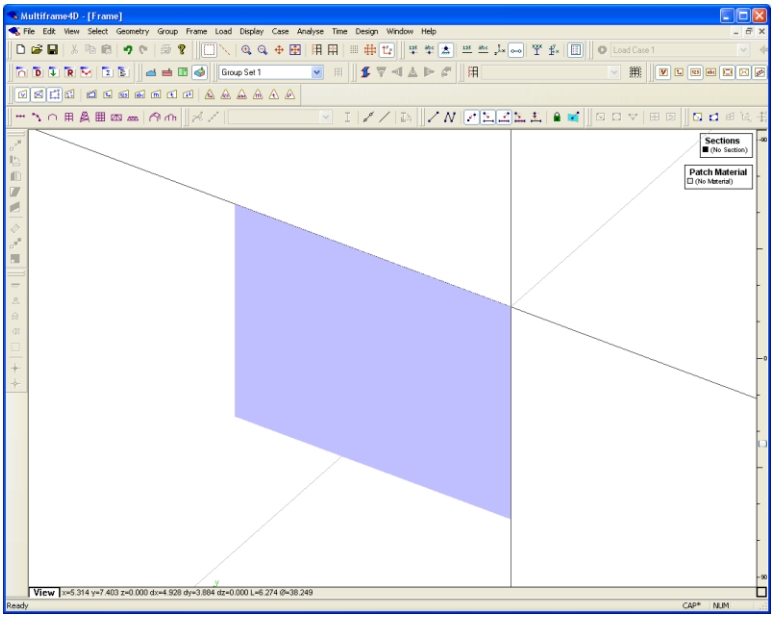


Adding a Patch

There are several ways to add a patch. The types of patches that can be added are rectangular patches, triangular patches, quadrilateral patches, and automatically created groups of patches in a floor or frame. The commands to add these are available from the Geometry menu or from the patch toolbar. They are similar to the corresponding commands for adding load panels.

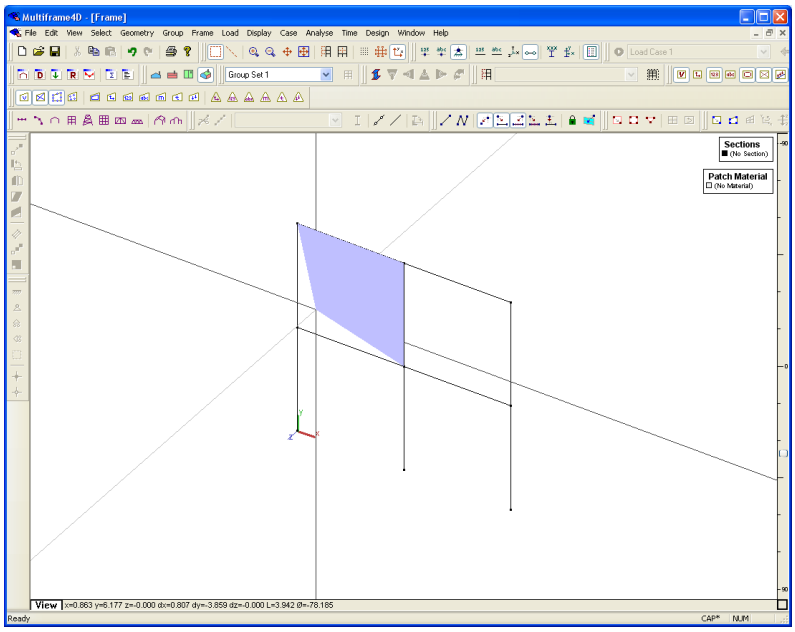
To add a rectangular patch

- **Choose Add rectangle Patch from the Geometry menu**
- **Click on two points/nodes at diagonally opposite corners of the patch.**



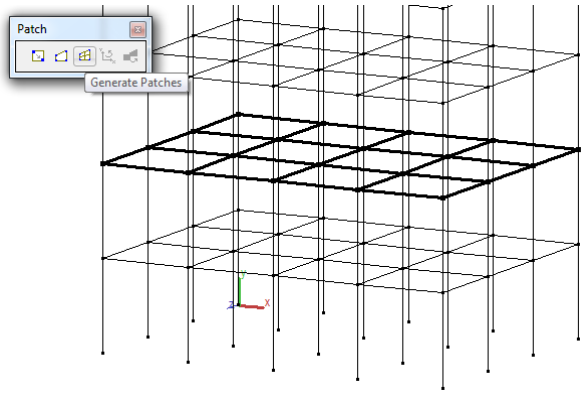
To add a quadrilateral patch

- **Choose Add N-node Patch from the Geometry menu**
- **Click in anti-clockwise order the four points or nodes that define the corners of the patch.**

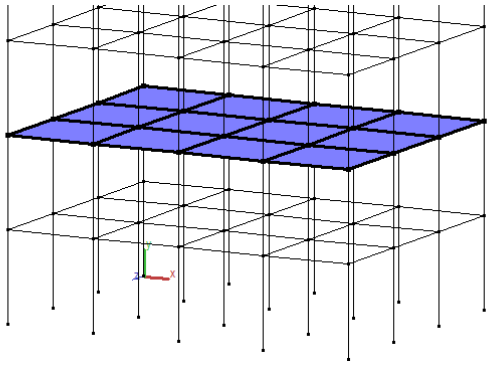


To automatically add Patches between all of the members in a planar region (eg a floor or frame)

- **Select the members which define the planar region**

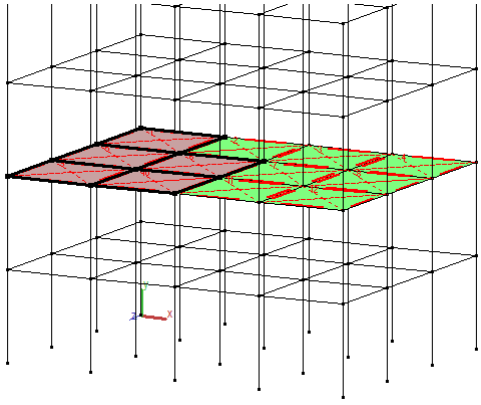


- **Choose Auto-generate Patches from the Geometry menu**

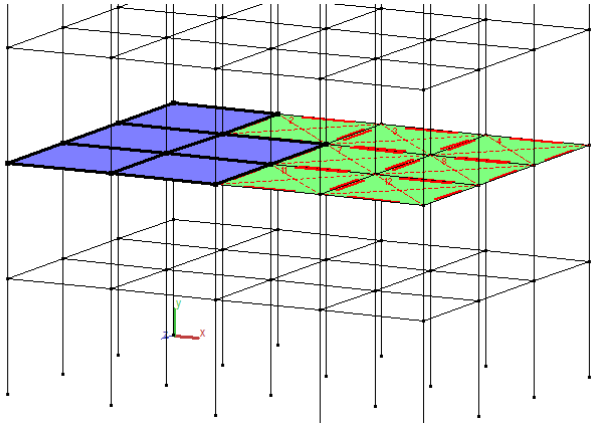


To automatically convert load panels to Patches

- **Select a group of load panels to be converted in the Frame window**



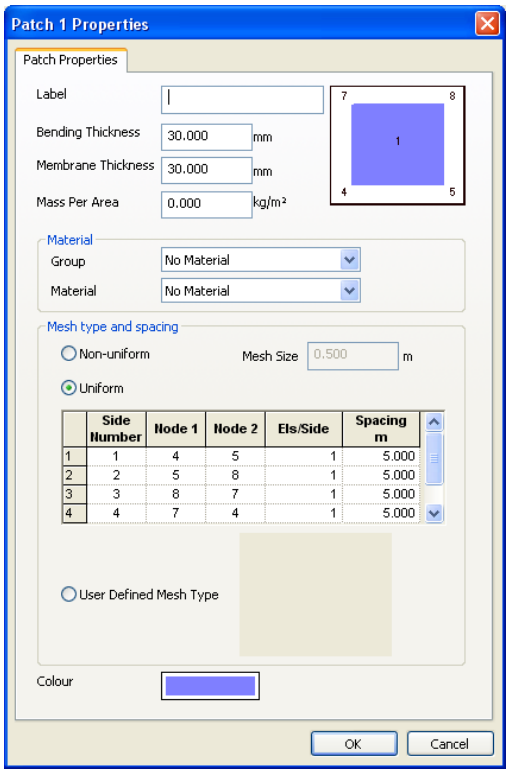
- **Choose Convert Load Panel to Patch from the Geometry menu**



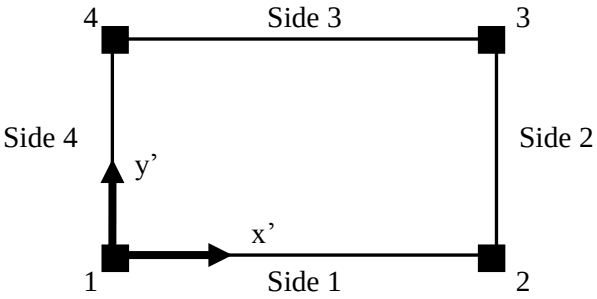
Editing a Patch

To change the properties of a patch

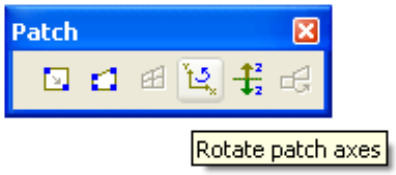
- **Double click on the patch in the Frame window**



To edit patch mesh densities, enter a label and thickness for the patch, and also edit the material and user colour of the patch. The numbers of the nodes and sides of the patch are as shown in the diagram below.



To edit the patch node sequence by using the Rotate Patch Axes icon in the Patch toolbar.



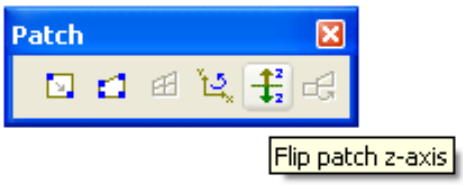
This will rotate the patch nodes so that they all move one place anti-clockwise. This is most useful when used several times to change the local axes and direction of a patch.

To change the axes of a group of patches

- **Select a group of patches to be changed in the Frame window**
- **Right click and choose Rotate Patch Axes from the pop up menu**
- **Click on an icon to change local axes of the patches**

The same technique is used to change the labels and colours of a group of patches by choosing the corresponding items form the pop up menu.

To flip patch normal direction by using the Flip Patch Axes icon in the Patch toolbar.



This will flip patch normal direction, i.e. z'-axis regardless of node sequences so that they all move all patch normal to their opposite directions. This is most useful when assigning all patches to the same normal direction.

To change the normal directions of a group of patches

- **Select a group of patches to be changed in the Frame window**
- **Right click and choose Flip Patch Axes from the pop up menu**
- **Click on an icon to change local axes of the patches**

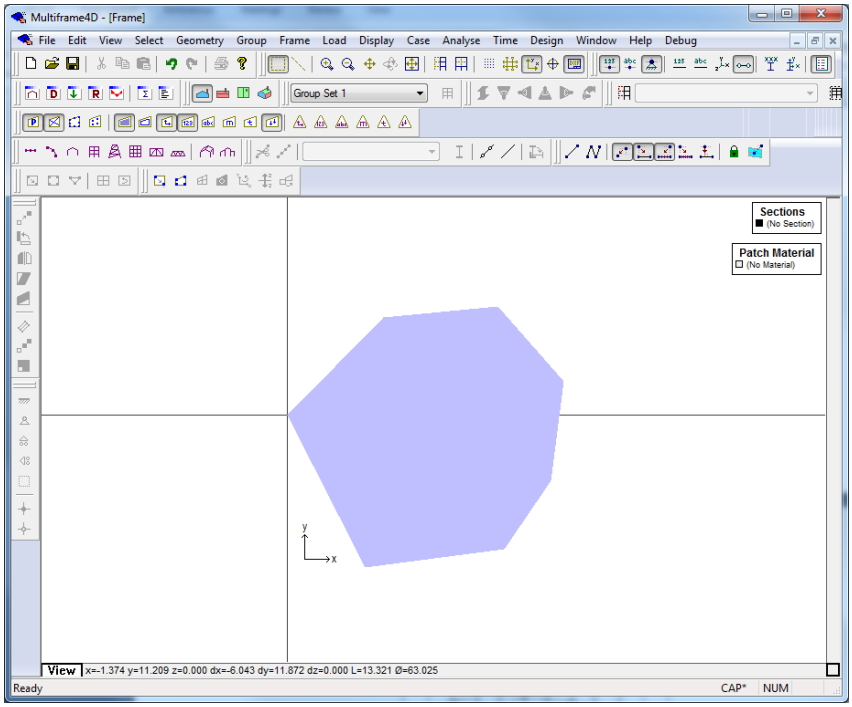
The same technique is used to change the labels and colours of a group of patches by choosing the corresponding items from the pop up menu.

Add N-sided Patches & Openings

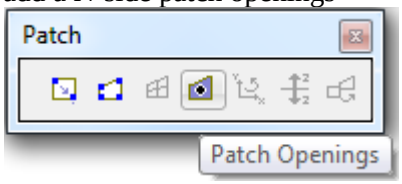
Multiframe allows n-side patches to be drawn in structural designing. Add N-node patch command is available from the Geometry menu or from the patch toolbar.

To add a N-side patch

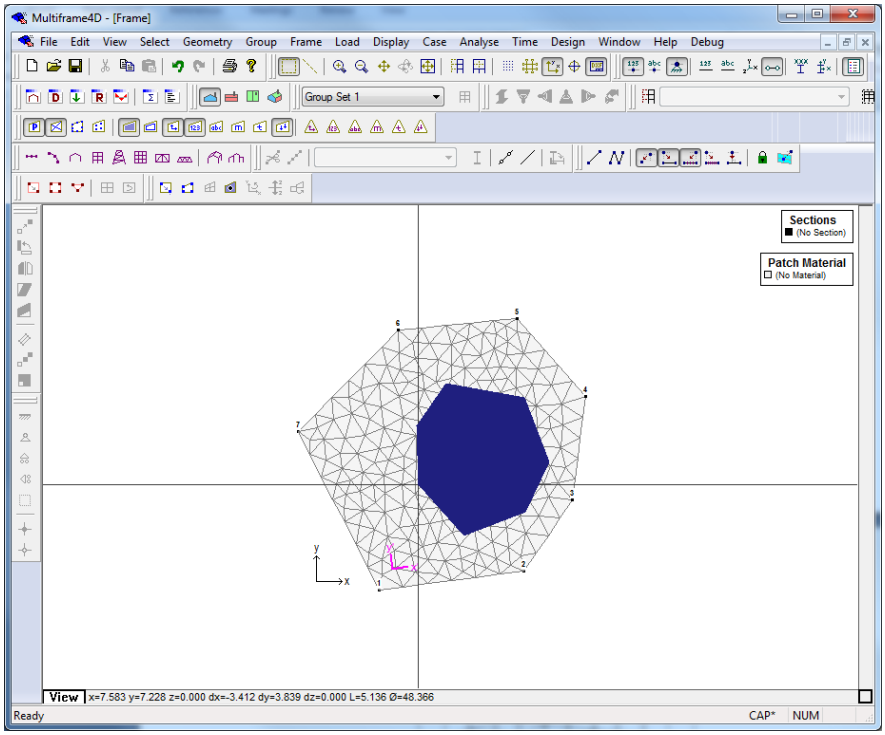
- **Choose Add n-node Patch from the Geometry menu**
- **Click in anti-clockwise order the N points or nodes that define the corners of the patch**



To add a N-side patch openings



- **Choose Add Patch Openings from the Geometry menu or icon button**
- **Click in anti-clockwise order the N points or nodes that define the corners of the patch opening**

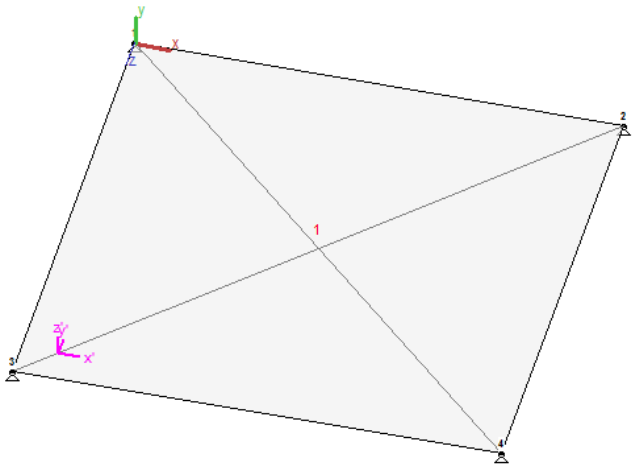


3D & Curved Patch Support

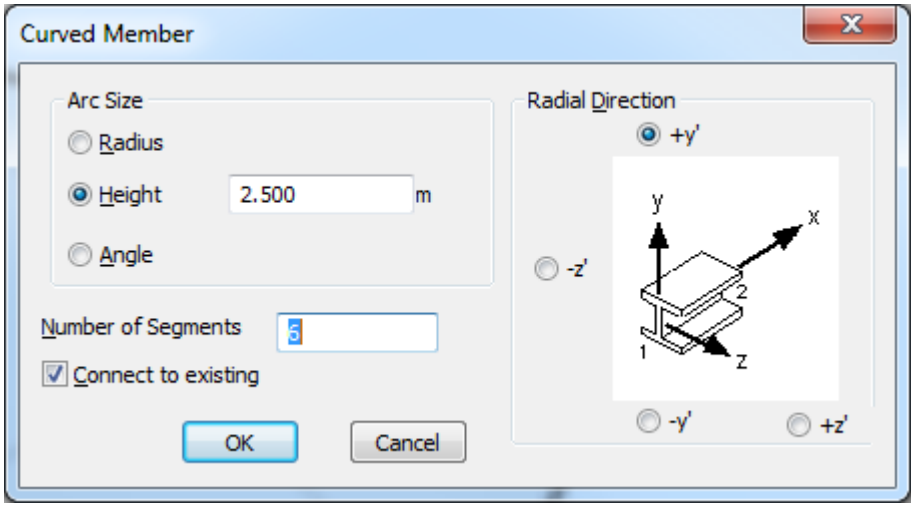
Multiframe allows 3D and/or curved patches to be created. Patch curved edges will be defined by its edge design members. Users can use the existing patch operation commands to create 3D and/or curved patches. For example,

To create a 3D & curved patch

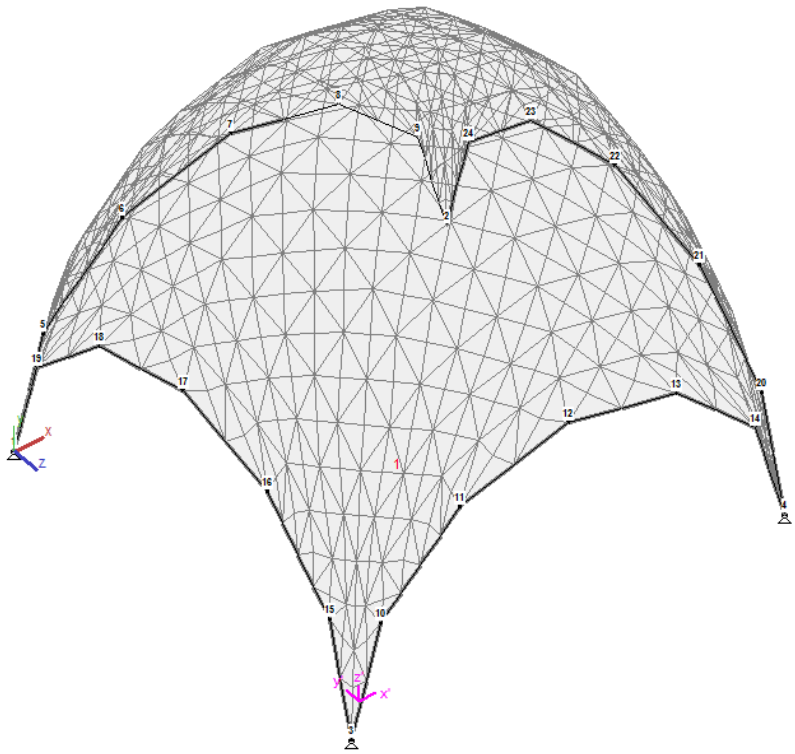
- **Create a planar patch with member on their sides**



-
- **Select side members, and choose "Convert member to Arc…" command**

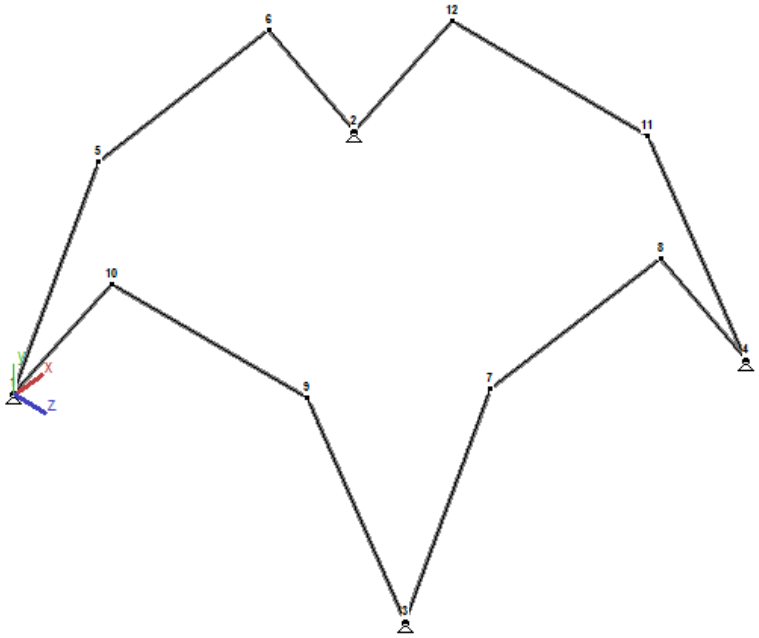


- **Click OK and remesh the patch to get**

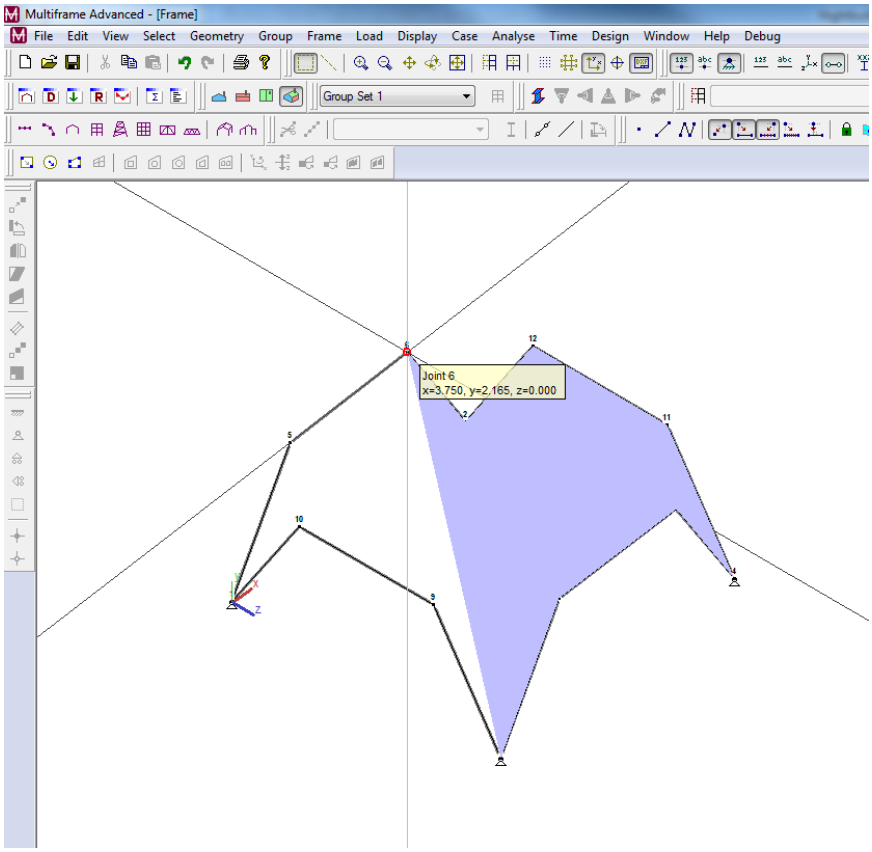


Users can also use “Add Patch” command to click on the a list of patch corner nodes to create a 3D patch

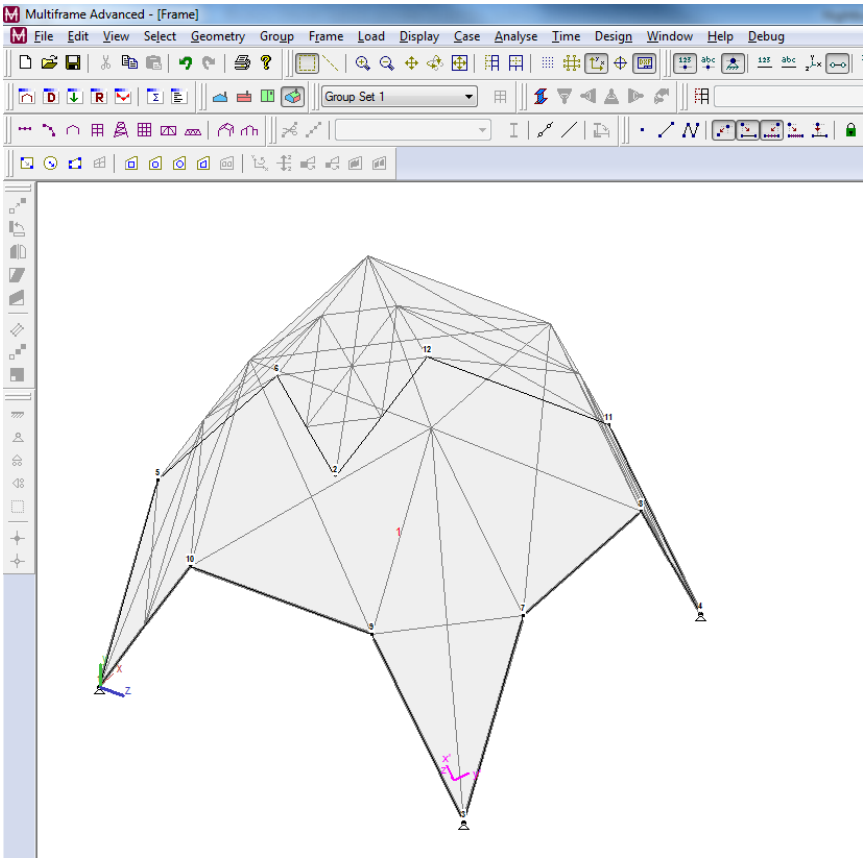
- **Create a list of patch corner nodes**



- **Sketch node to node of the above patch corners**


Multiframe Advanced - [Frame]
File Edit View Select Geometry Group Frame Load Display Case Analyse Time Design Window Help Debug
Group Set 1
Joint 6
x=3.750, y=2.165, z=0.000

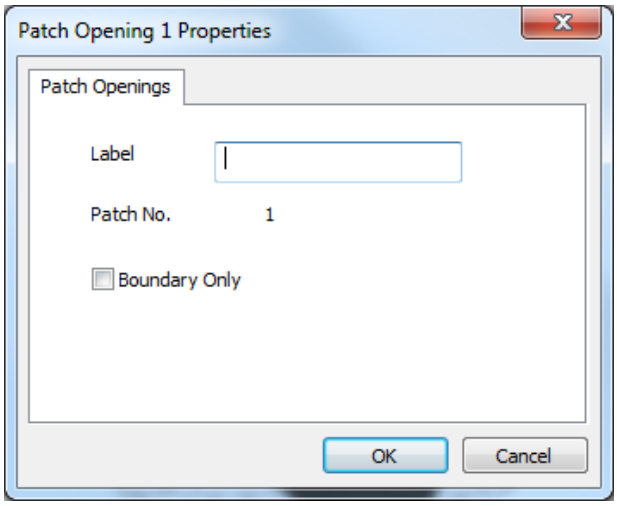
- **Mesh the patch and get**


Multiframe Advanced - [Frame]
File Edit View Select Geometry Group Frame Load Display Case Analyse Time Design Window Help Debug
Group Set 1

Edit Patch Openings

To change the properties of a patch opening

- **Double click on the patch opening in the Frame window**



You can enter a label. If boundary only option is selected, Multiframe will add triangular mesh onto the area of openings, but as well as retain the opening boundaries. This option will be very useful for users to increase the mesh densities of certain areas of a patch or retain a line segment or members inside a patch.

Delete Patch Openings

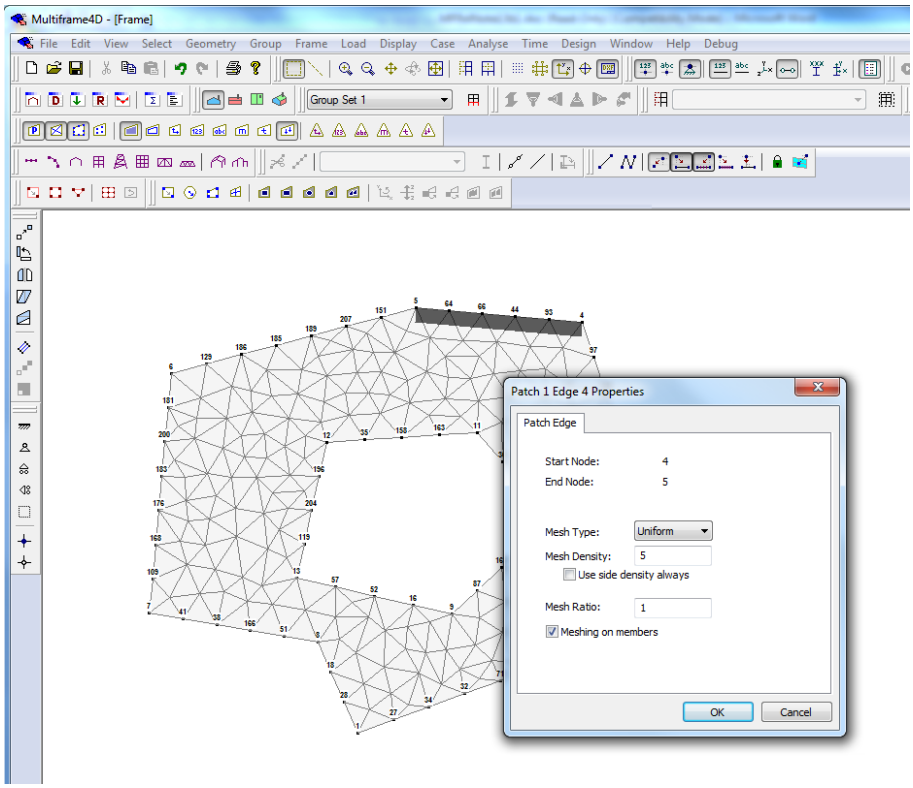
To delete a patch opening

- **Select patch openings in the Frame window**
- **Hit the delete key from key board or Choose Delete Patch Openings from the Geometry menu**

Edit Patch/Opening Edges

To change the properties of a patch/opening edge

- **Double click on the patch/opening edge in the Frame window**



You can change the mesh type of a patch/opening edge, mesh density and mesh ratio etc. You can also decide whether you would like patch/opening edge members to be meshed or not. Patch/opening edge option will be very useful for users to interact with mesh density along the edges to their adjacent patches.

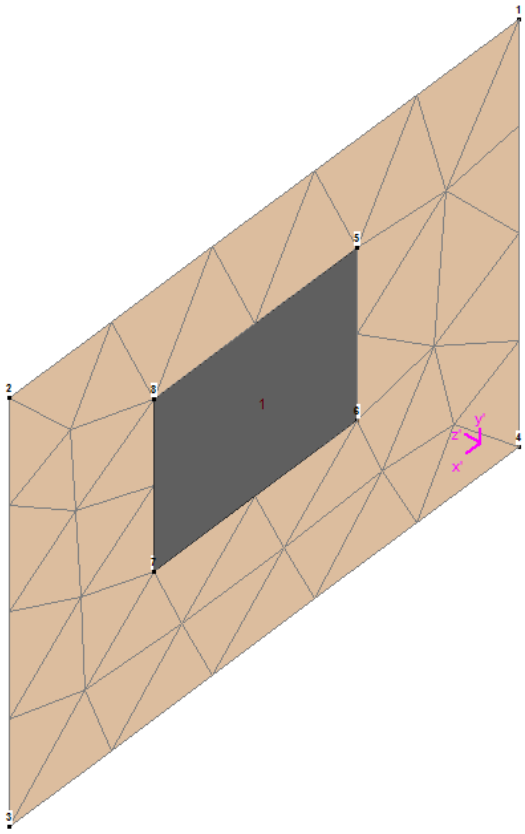
Extrude Rider Bar

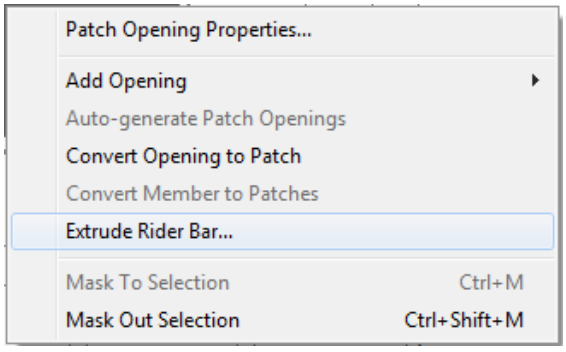
Multiframe allows patches to be created surrounding a patch opening like a rider bar in marine design. Extrude rider bar command is available from the Geometry menu or from the patch opening context menu.

To extrude rider bar of a patch opening

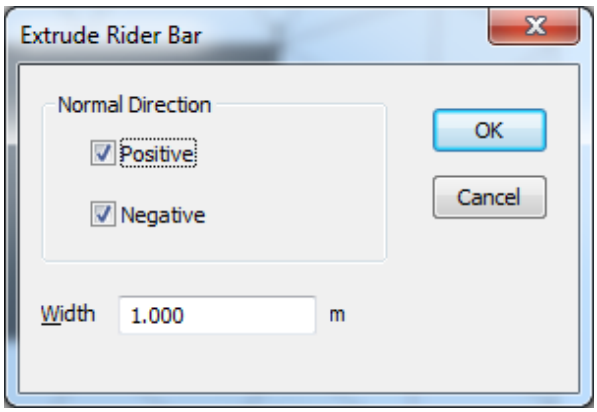
- **Select patch openings in the Frame window, and mouse right click**

- **Choose Extrude Rider Bar…**

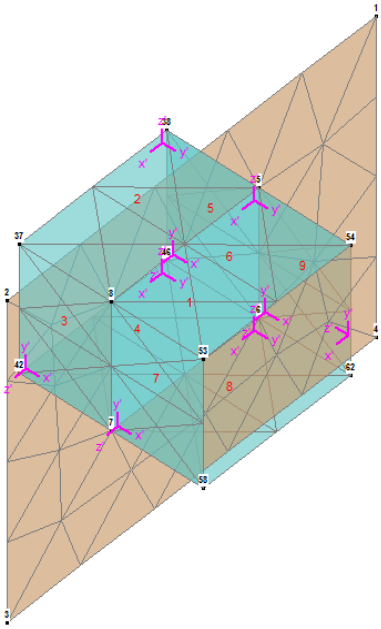




- **Edit extrude properties**



- **Click OK**



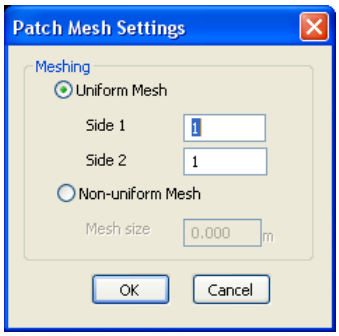
Delete Patch/Opening Edges

Either patch or opening edges cannot be deleted unless their parent patches or openings have been deleted.

Meshing a Patch

To change the mesh densities of patches

- **Double click on the patch in the Frame window or Select a group of patches to be changed in the Frame window**
- **Right click and choose Patch Meshing from the pop up menu**



For uniform meshes specify the mesh density along each side of the patch. For non-uniform meshes type in a value for maximum length of a plate side and Multiframe automatically generates all plates for you.

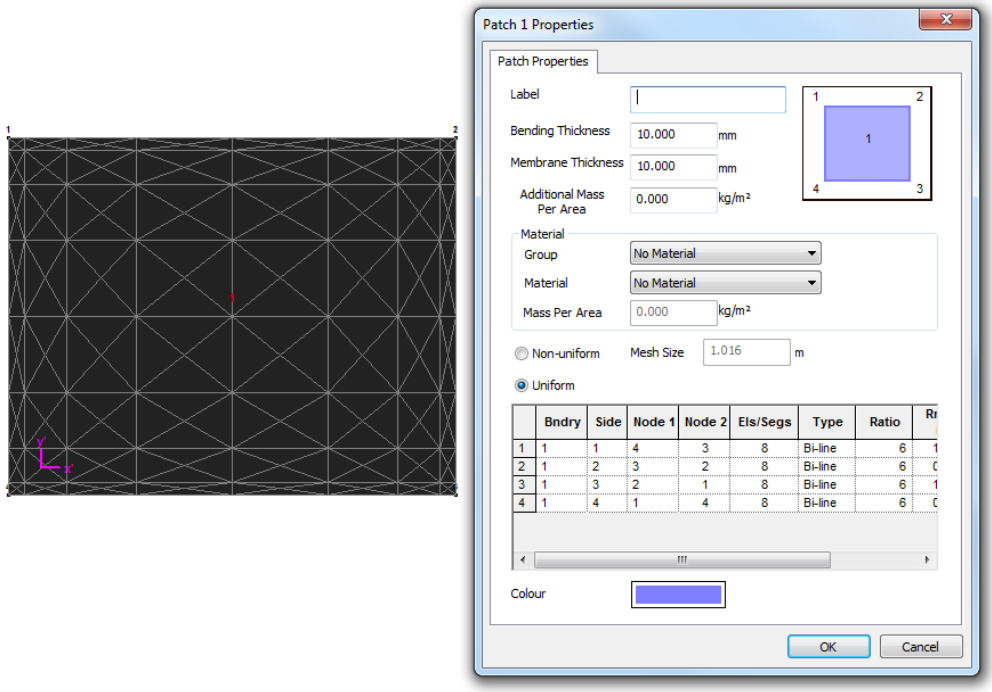
Patch Mesh Patterns

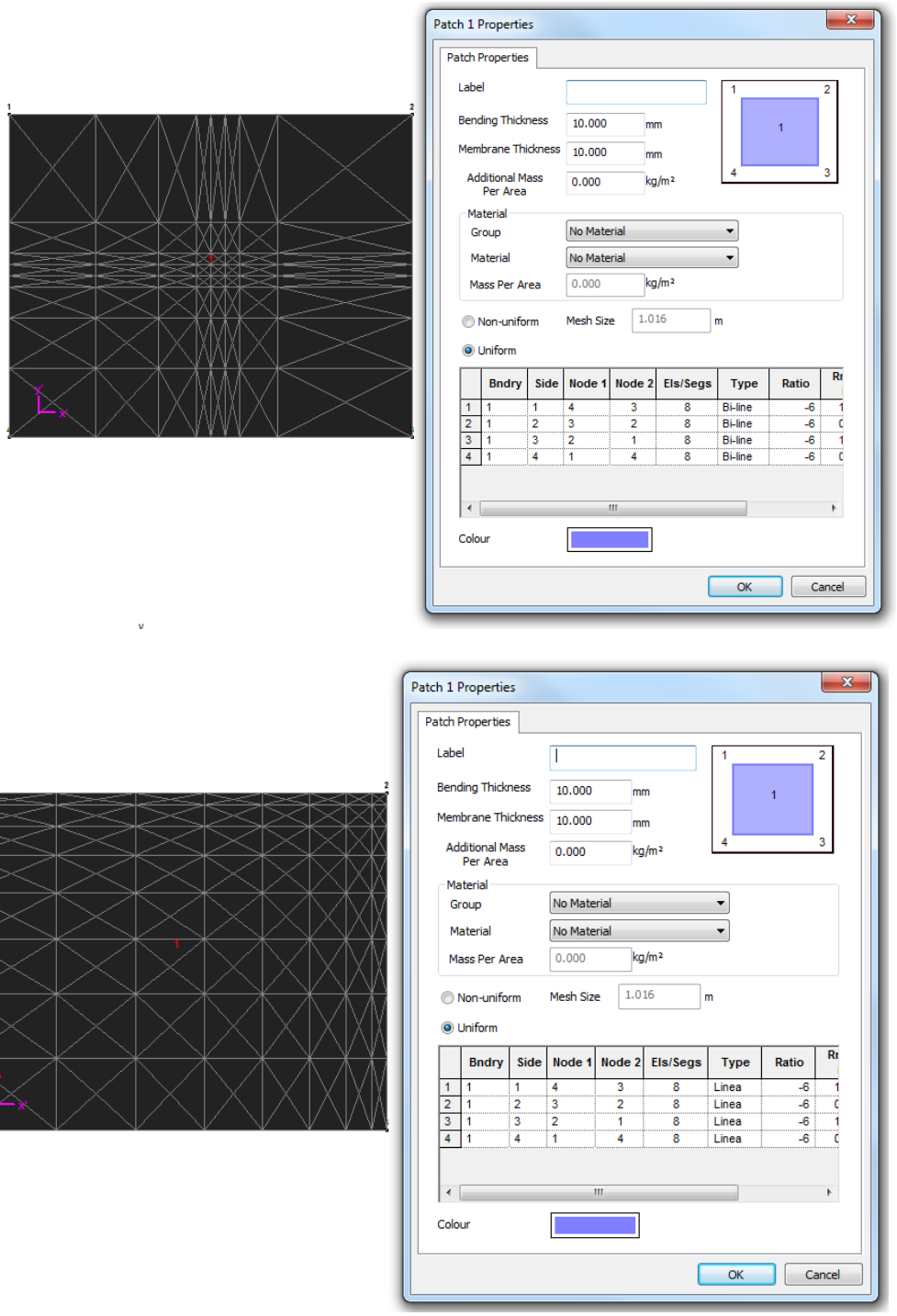
There are two types of triangular meshes for patches, uniform and non-uniform. 3- or 4-sided patches without openings could be meshed to uniform patterns. All other patches with or without openings have to be non-uniformly meshed.

At each patch or opening side, there are three mesh types, uniform, linear and bi-linear. To mesh a patch, you could specify the mesh density, type and ratio along each side of the patch and openings. In patch property window, the user defined mesh type option has been removed, replacing with a comprehensive patch side meshing algorithm. Each segment/side has new fields such as density, type and ratio.

- Density – number of segments of a patch/opening side;
- Type – mesh type can be uniform, linear or bi-linear;
- Ratio – is the length proportion between max sub-segment (Rmax) and the min sub-segment (Rmin). If Ratio is negative value, meshing along the patch/opening side will be in an opposite direction.

With different combinations among density, type and ratio, users could create all kinds of patch mesh patterns.





Displaying a Patch or Plate

Multiframe allows control of the display of a range of symbols associated with patches and plates. These options are available via the Symbols command in the Display menu or more commonly via the icons in the Patch-Plate Symbols toolbar.

These icons control visibility of

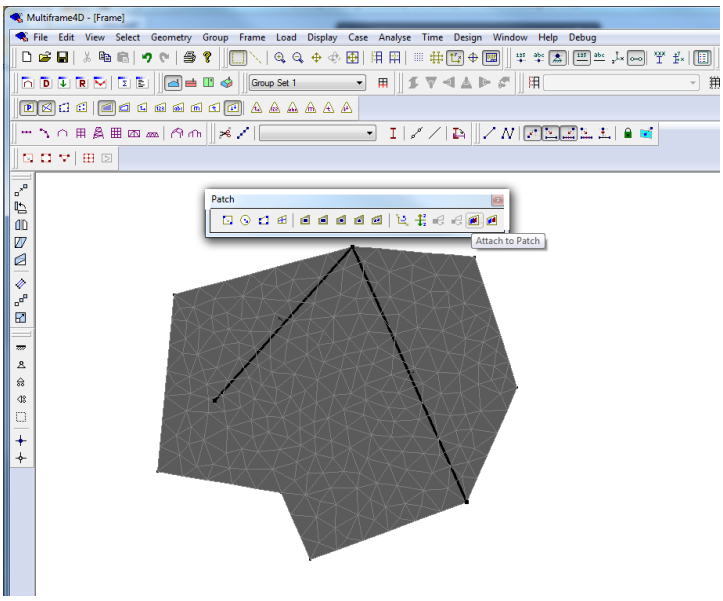
- **patches, patch meshes, edge nodes, internal nodes**
- **patch colour fill, patch outline, patch axes, patch numbers, patch labels, patch materials patch thickness, patch loads**
- **plate axes, plate numbers, plate labels, plate materials, plate thickness, plate loads**

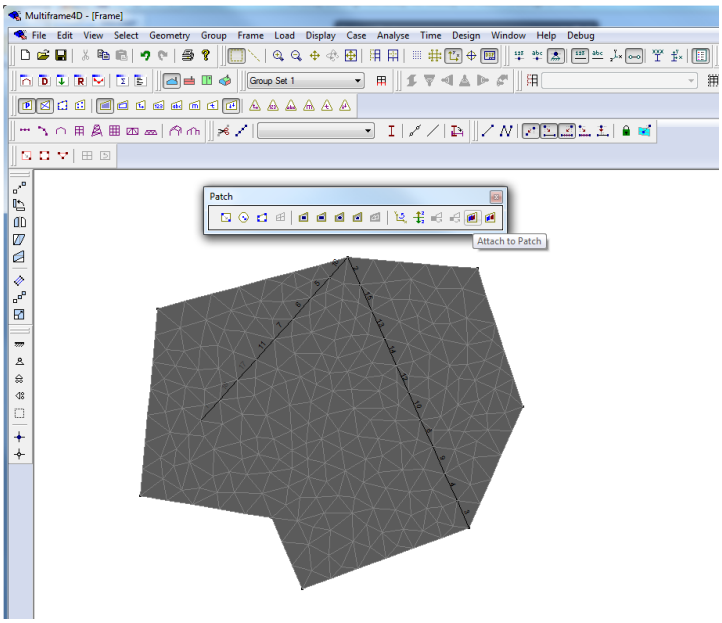
Attach Member/Patch to Patch

Multiframe allows members/patches to be attached to patches in structural designing, so that those attached members or attached patch edges will be forced to mesh during patch meshing. Attach to Patch commands is available from the Geometry menu or from the patch toolbar.

To attach a member/patch to patch

- **Select members and/or patches**
- **Choose Attach to Patch Command from the Geometry menu or icon button**


Multiframe4D - [Frame]
File Edit View Select Geometry Group Frame Load Display Case Analyse Time Design Window Help Debug
Group Set 1
Patch
Attach to Patch

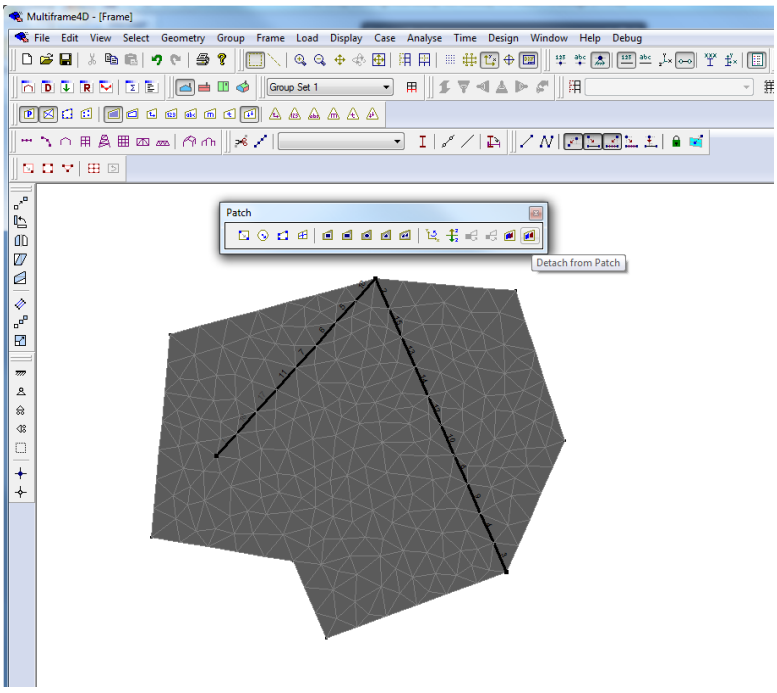

Multiframe4D - [Frame]
File Edit View Select Geometry Group Frame Load Display Case Analyse Time Design Window Help Debug
Group Set 1
Patch
Attach to Patch

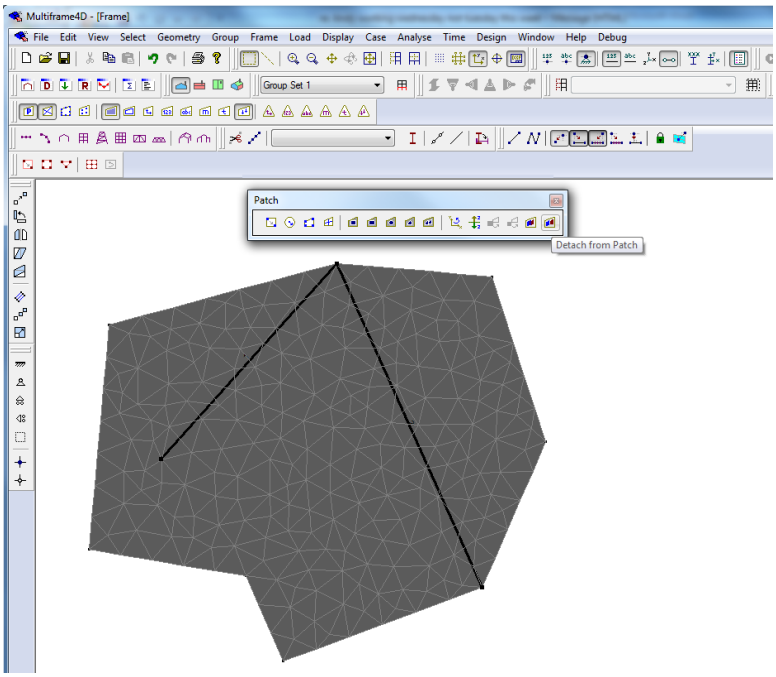
Detach Member/Patch to Patch

Multiframe allows members/patches to be detached from patches in structural designing, so that those detached members or patch edges will not be meshed during patch meshing. Detach from Patch commands is available from the Geometry menu or from the patch toolbar.

To detach a member/patch to patch

- **Select members and patches that wish to be detached each other**
- **Choose Detach from Patch Command from the Geometry menu or icon button**


Multiframe4D - [Frame]
File Edit View Select Geometry Group Frame Load Display Case Analyse Time Design Window Help Debug
Group Set 1
Patch
Detach from Patch

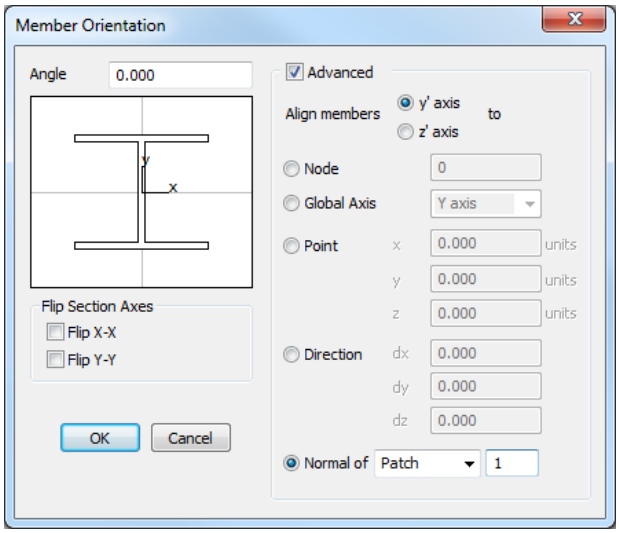

Multiframe4D - [Frame]
File Edit View Select Geometry Group Frame Load Display Case Analyse Time Design Window Help Debug
Group Set 1
Patch
Detach from Patch

Align Member to Patch or Load Panel

It is possible to align a member so that it is perpendicular to the patch that it is attached to.

- **Select one or more members**
- **Choose Member Orientation… from the Frame menu**

The following dialog will be displayed



From the Member Orientation dialog

- **Select Advanced**
- **Select Normal of Patch/Load Panel**
- **Choose Patch or Load Panel from the drop down box**

- **Enter the Patch**

If you select a patch or load panel as well as a member/s before opening the Member Orientation dialog, these last fields will already be populated. It is still possible for you to change them.

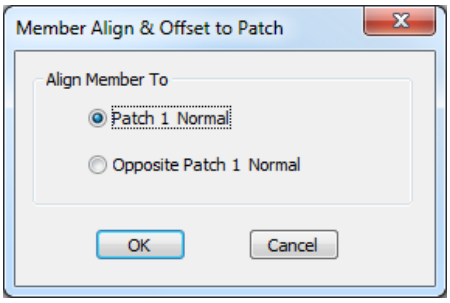
The member will be orientated perpendicular to the patch on the same side as the patch normal. If you wish to align the member opposite the patch normal, then do the above and then click Flip X-X and the section will be rotated by 180 degrees. Any offset that the member has will also be reversed.

Automatically Align and Offset Member to Patch/Load Panel

The Member Align and Offset to Patch/Load Panel command will automatically align a member to be perpendicular to a patch or load panel. An offset is applied so that the bottom of the section is flush with the patch.

- **Select one or more members that are attached to**
- **Choose Member Align and Offset to Patch/Load Panel… from the Frame menu**

The following dialog will appear,



Select the side of the patch/load panel to orientate the member to.

If more than one patch/load panel is selected, the selected members will be aligned to the first patch/load panel (ie lowest number). If a patch and a load panel is selected, the members will be aligned to the patch.

Please note that the member will be offset to its base plus half the bending thickness of the patch. In the render mode the patch thickness is not drawn so the member will appear a small distance from the patch. Load panels do not have a thickness.

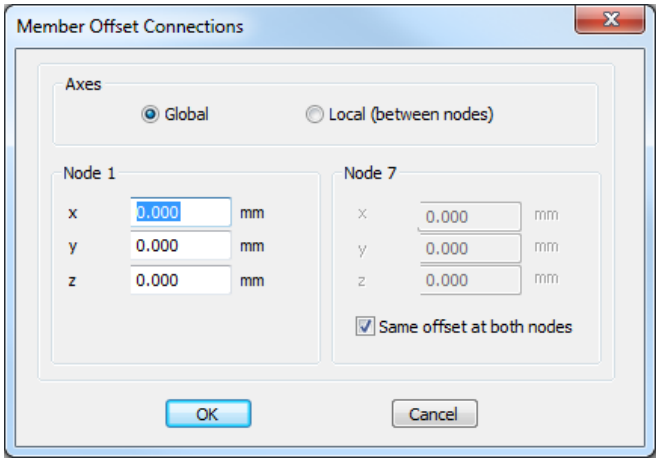
Member Offset Connections

The member offset connection dialog has been improved.

To open the Member Offsets dialog,

- **Select one or more members**
- **Choose Member Offsets… from the Frame menu**

The following dialog will be displayed



The Node labels now display the actual node numbers for each end of the selected member.

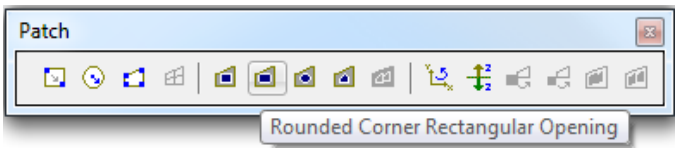
If you tick the Same offset at both nodes checkbox the offsets at the second node will be auto-completed.

Add Rounded Corner Rectangular Opening

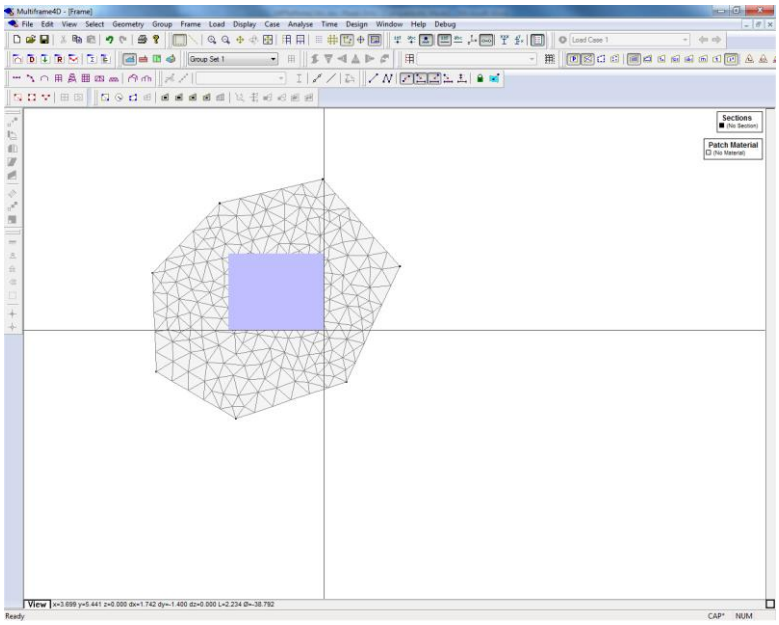
Multiframe allows rounded corner rectangular openings to be drawn in structural designing. Add rounded corner rectangular opening command is available from the Geometry menu or from the patch toolbar.

To add a rounded corner rectangular opening

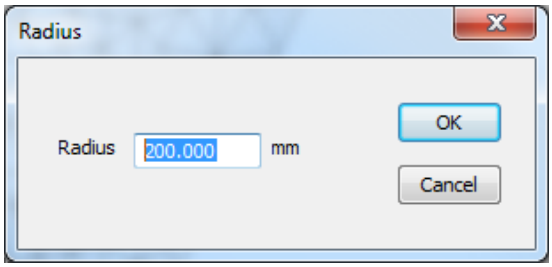
- **Choose Add Opening → Rounded Corner Rectangular from the Geometry menu or icon button**



- **Click two points or nodes that define the corners of the rectangular opening**



- **Input corner radius and click OK button**



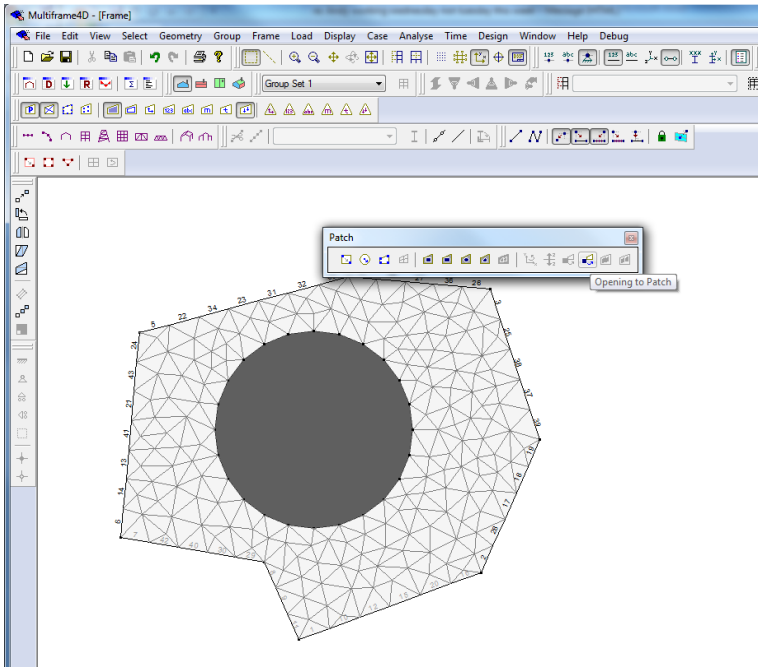
-

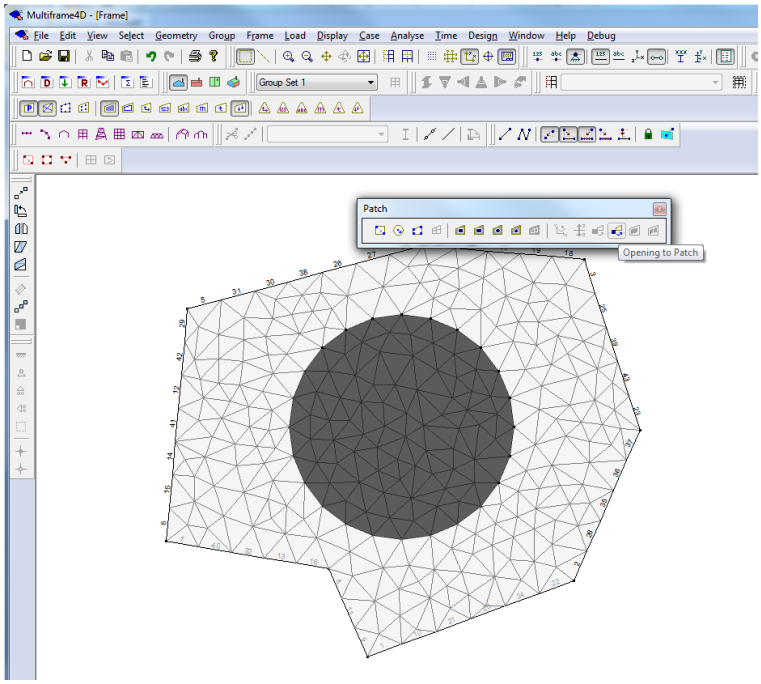
Convert Opening to Patch

Multiframe allows openings to be converted to patches in structural designing based on the selections of openings. Convert Opening to Patch commands is available from the Geometry menu or from the patch toolbar.

To convert openings to patches

- **Select openings**
- **Choose Convert Opening to Patch Command from the Geometry menu or icon button**


Multiframe4D - [Frame]
File Edit View Select Geometry Group Frame Load Display Case Analyse Time Design Window Help Debug
Group Set 1
Patch
Opening to Patch

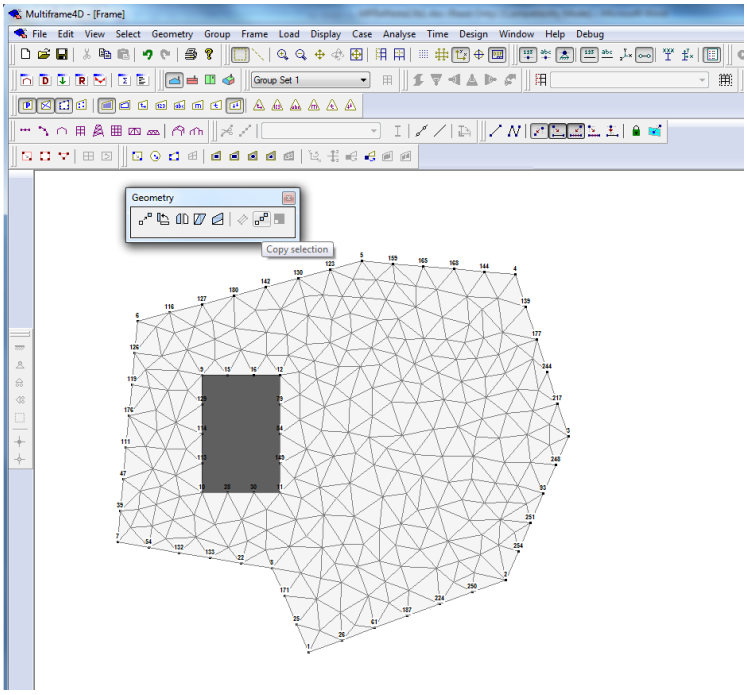

Multiframe4D - [Frame]
File Edit View Select Geometry Group Frame Load Display Case Analyse Time Design Window Help Debug
Group Set 1
Patch
Opening to Patch

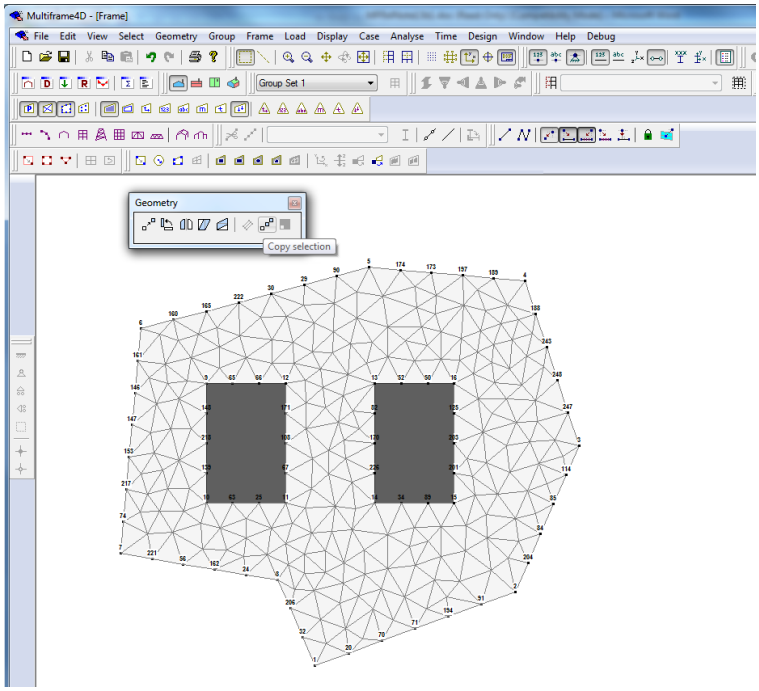
Opening Duplicate

Multiframe allows openings to be copied in structural designing. Duplicate command is available from the Geometry menu or from the Geometry toolbar.

To duplicate an opening

- **Select openings**
- **Choose Duplicate from the Geometry menu or icon button**


Multiframe4D - [Frame]
File Edit View Select Geometry Group Frame Load Display Case Analyse Time Design Window Help Debug
Group Set 1
Geometry
Copy selection

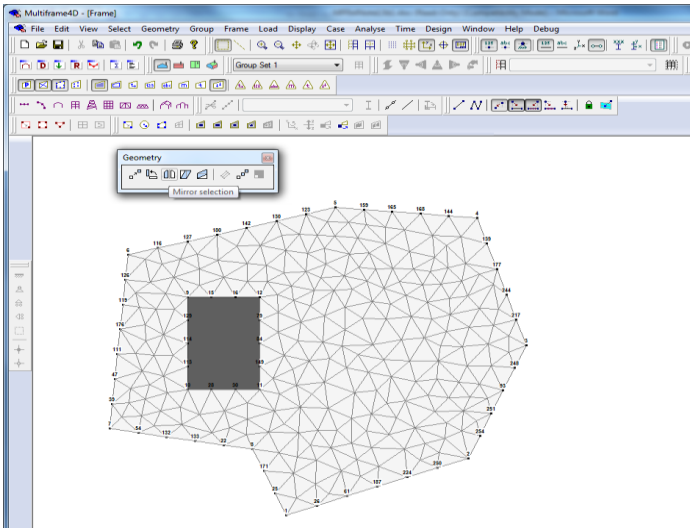

Multiframe4D - [Frame]
File Edit View Select Geometry Group Frame Load Display Case Analyse Time Design Window Help Debug
Group Set 1
Geometry
Copy selection

Opening Mirror

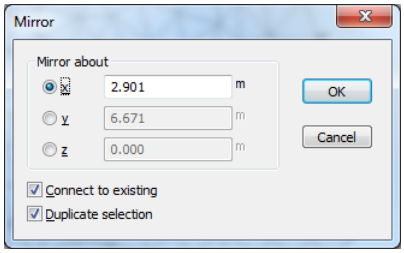
Multiframe allows openings to be mirrored in structural designing. Mirror command is available from the Geometry menu or from the Geometry toolbar.

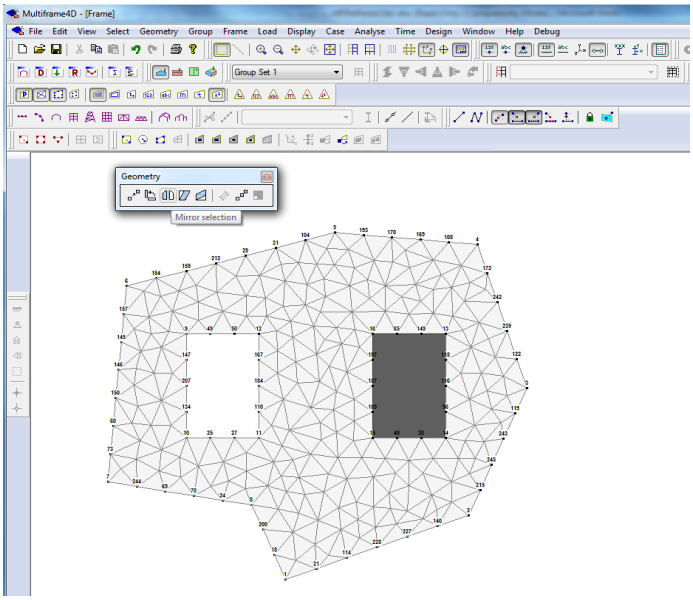
To mirror an opening

- **Select openings**
- **Choose Mirror from the Geometry menu or icon button**



- **Draw or input mirror axes**



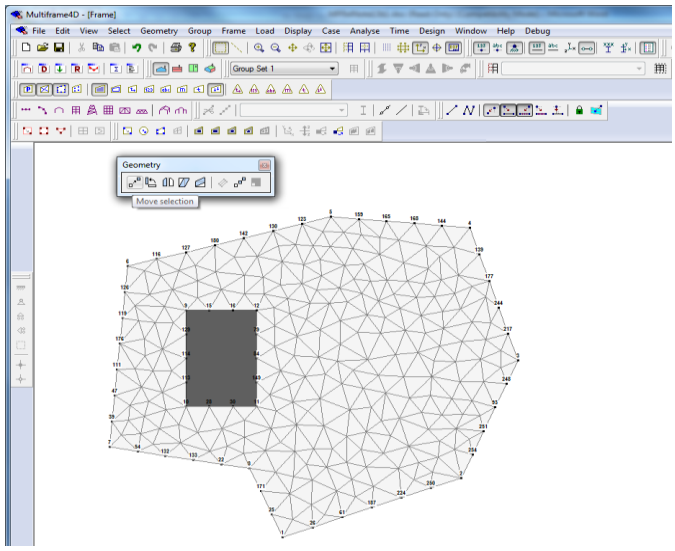


Opening Move

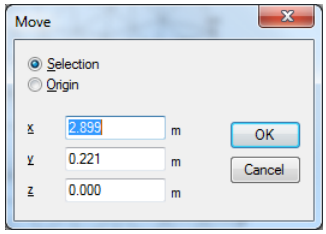
Multiframe allows openings to be moved in structural designing. Move command is available from the Geometry menu or from the Geometry toolbar.

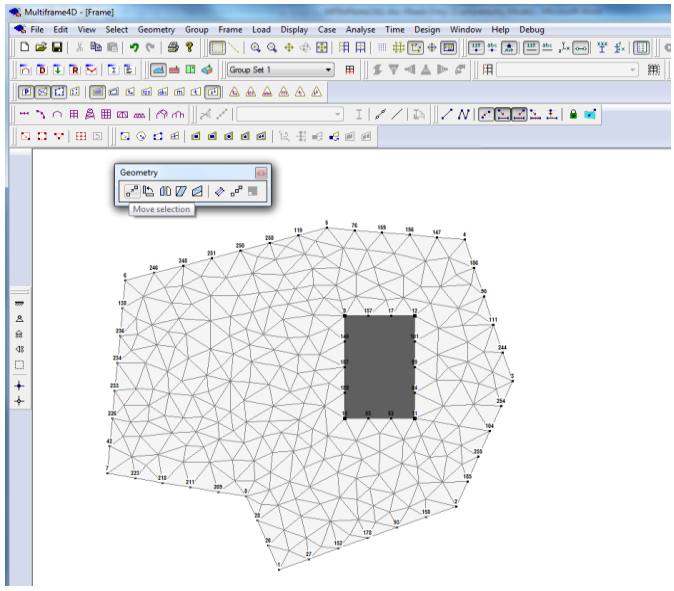
To move an opening

- **Select openings**
- **Choose Move from the Geometry menu or icon button**



- **Draw or input move location, Click OK button**



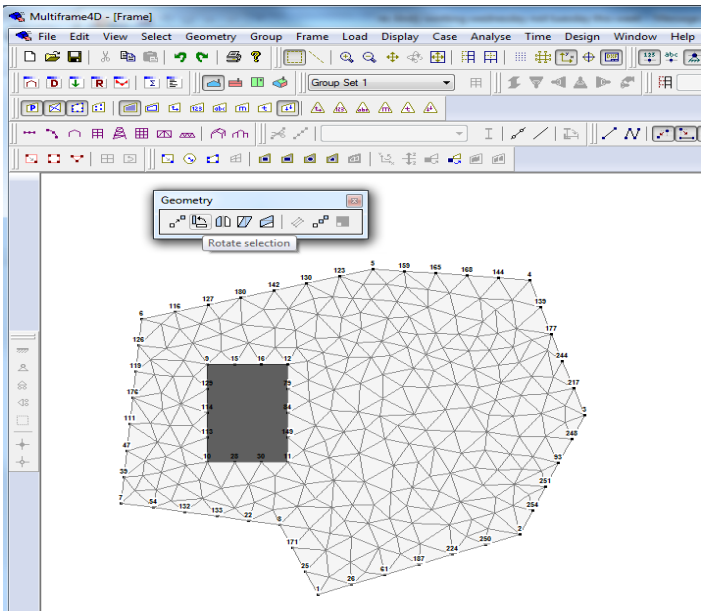


Opening Rotate

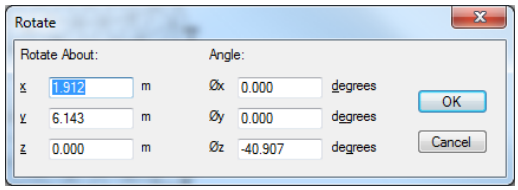
Multiframe allows openings to be rotated in structural designing. Rotate command is available from the Geometry menu or from the Geometry toolbar.

To rotate an opening

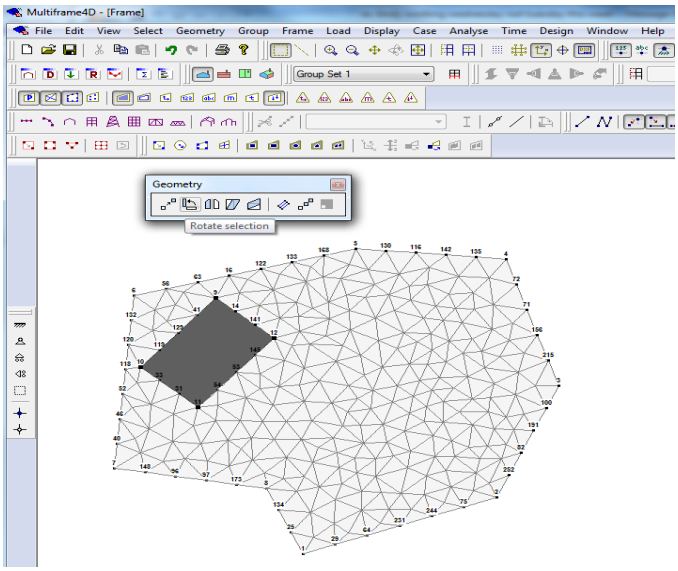
- **Select openings**
- **Choose Rotate from the Geometry menu or icon button**


Multiframe4D - [Frame]
File Edit View Select Geometry Group Frame Load Display Case Analyse Time Design Window Help
Group Set 1
Geometry
Rotate selection

- **Draw or input rotate axes and angle, Click OK button**



-

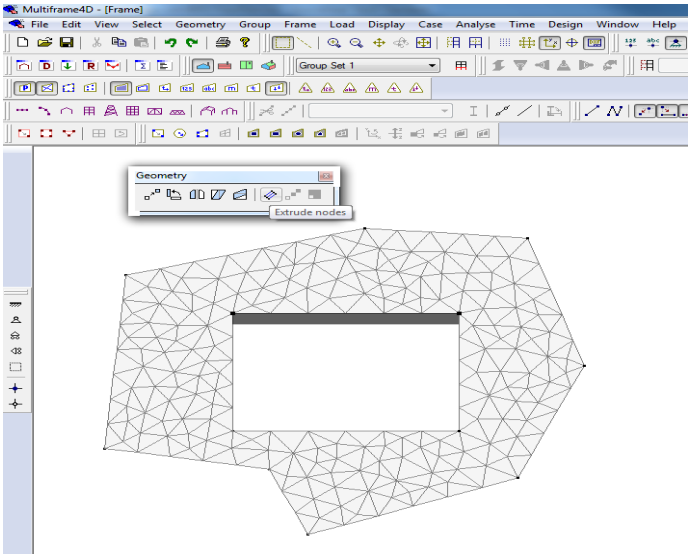

Multiframe4D - [Frame]
File Edit View Select Geometry Group Frame Load Display Case Analyse Time Design Window Help
Group Set 1
Geometry
Rotate selection

Patch/Opening Edge Extrude

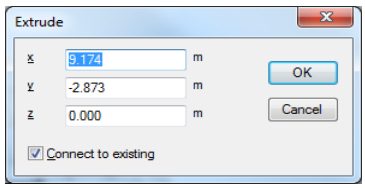
Multiframe allows both patch edges and opening edges to be extruded in structural designing. Extrude command is available from the Geometry menu or from the Geometry toolbar.

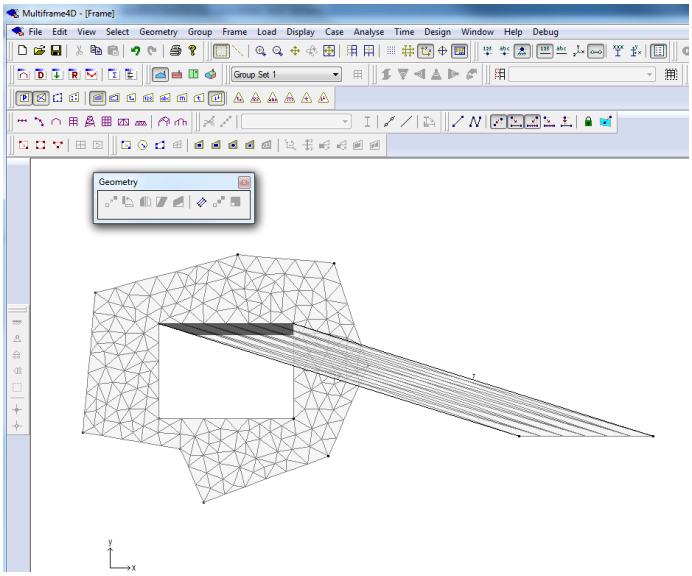
To extrude a patch/opening edge

- **Select the patch/opening edge**
- **Choose Extrude from the Geometry menu or icon button**


Multiframe4D - [Frame]
File Edit View Select Geometry Group Frame Load Display Case Analyse Time Design Window Help
Group Set 1
Geometry
Extrude nodes

- **Draw or input extrude axes, Click OK button**



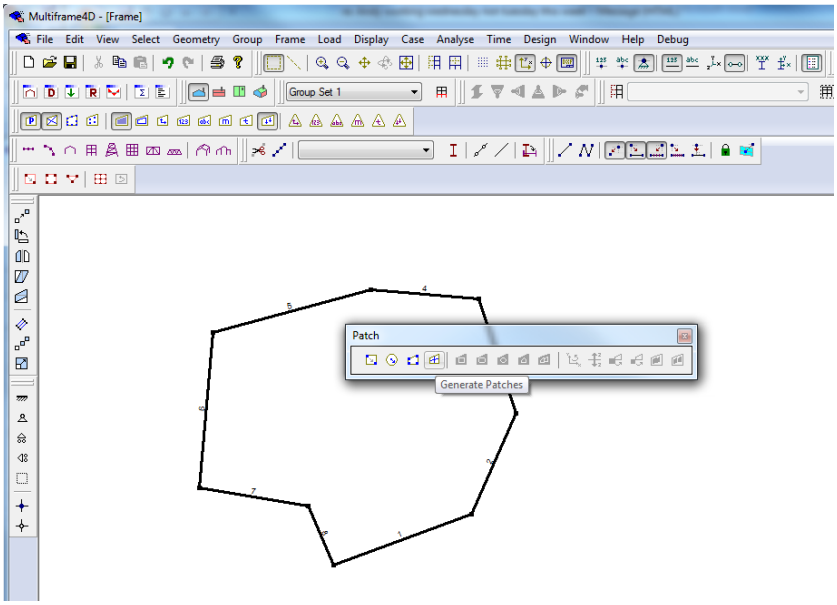

Multiframe4D - [Frame]
File Edit View Select Geometry Group Frame Load Display Case Analyse Time Design Window Help Debug
Group Set 1
Geometry

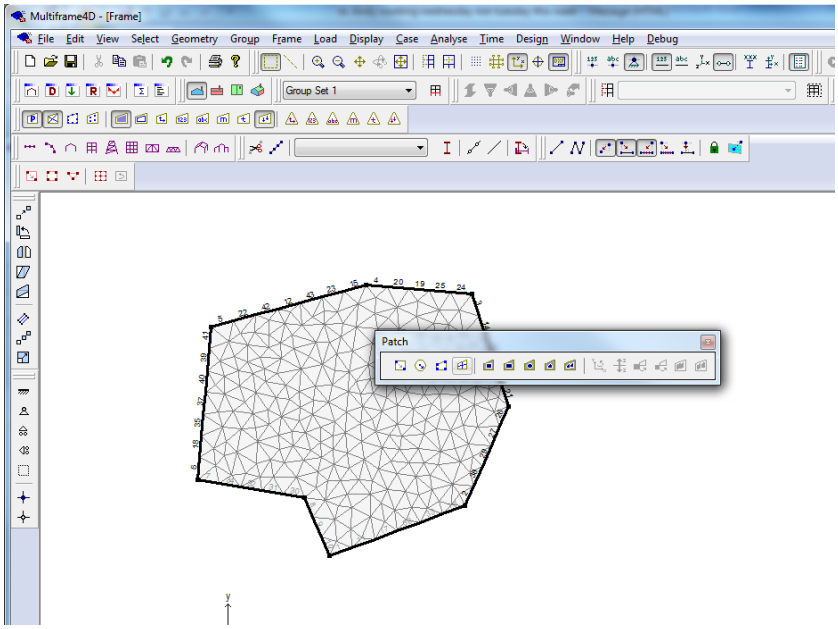
Generate N-sided Patch & Opening

Multiframe allows N-sided patches & openings to be generated in structural designing based on the member selections. Auto-generate Patches & Auto-generate Openings commands are available from the Geometry menu or from the patch toolbar.

To generate a N-sided patch

- **Select a list of members that form a N-sided patch**
- **Choose Auto-generate Patches from the Geometry menu or icon button**

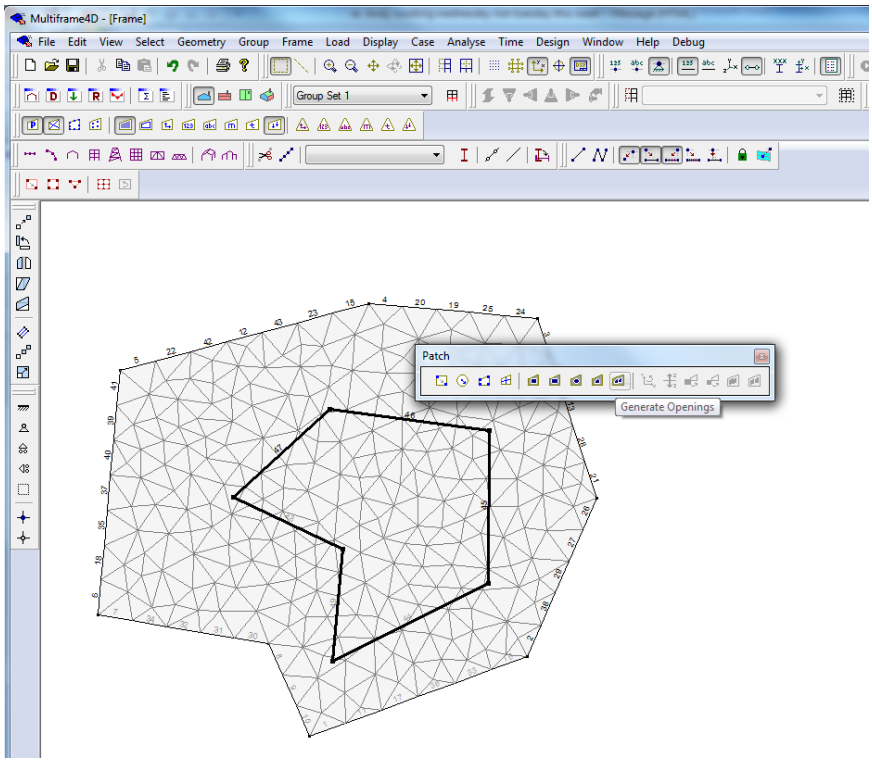

Multiframe4D - [Frame]
File Edit View Select Geometry Group Frame Load Display Case Analyse Time Design Window Help Debug
Group Set 1
Patch
Generate Patches

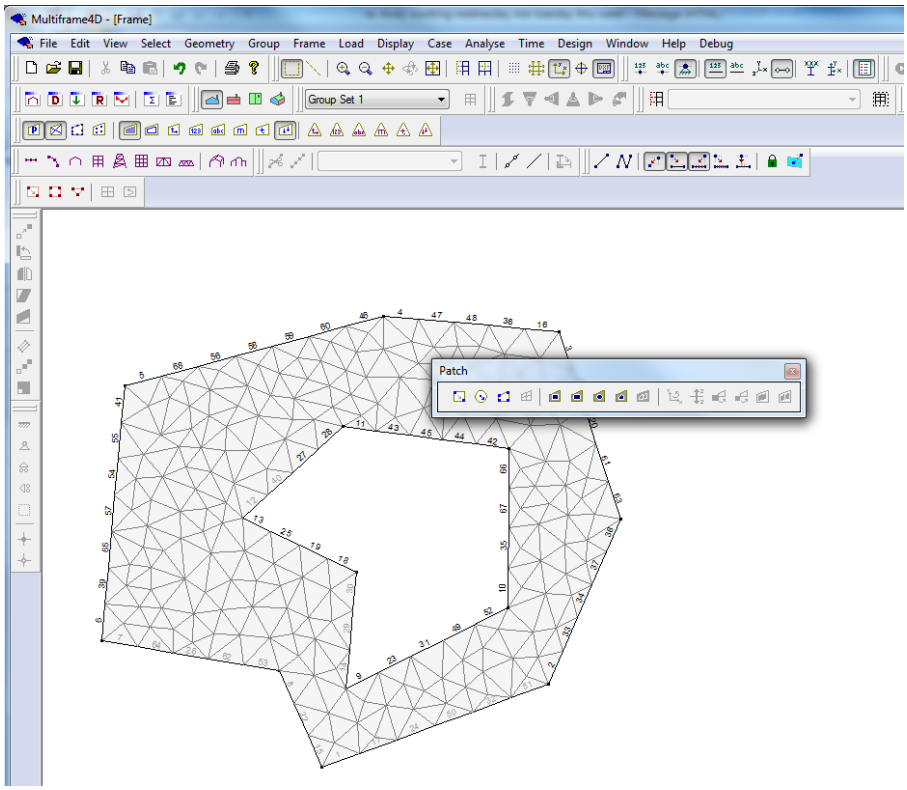

Multiframe4D - [Frame]
File Edit View Select Geometry Group Frame Load Display Case Analyse Time Design Window Help Debug
Group Set 1
Patch

➢

To generate a N-sided opening

- **Select a list of members that form a N-sided opening**
- **Choose Auto-generate Openings Command from the Geometry menu or icon button**


Multiframe4D - [Frame]
File Edit View Select Geometry Group Frame Load Display Case Analyse Time Design Window Help Debug
Group Set 1
Patch
Generate Openings

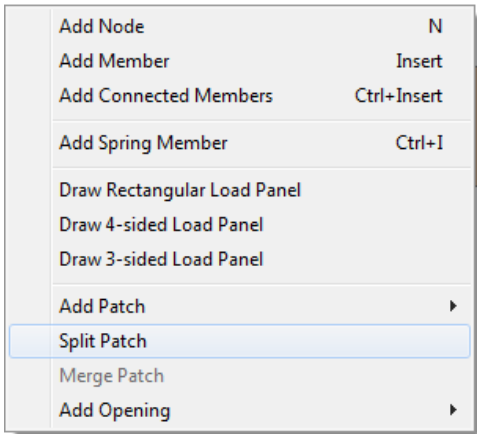

Multiframe4D - [Frame]
File Edit View Select Geometry Group Frame Load Display Case Analyse Time Design Window Help Debug
Group Set 1
Patch

Patch Split

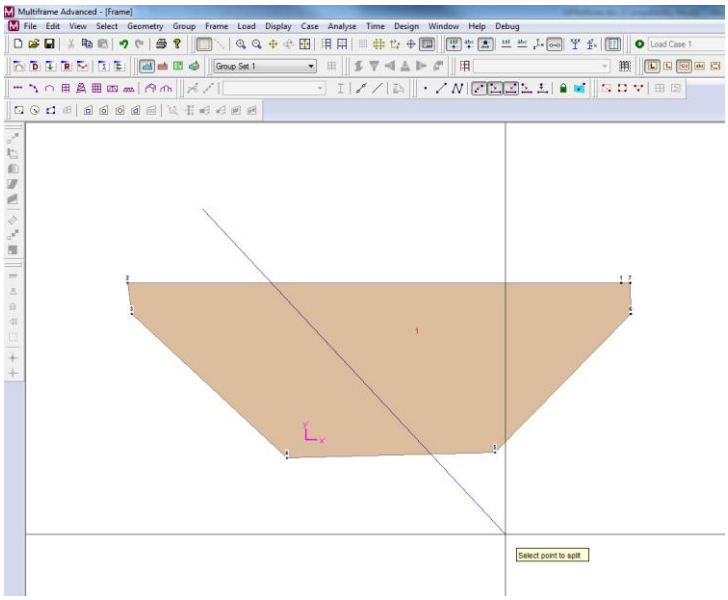
Multiframe allows patches to be split into multiple sub-patches. Patch split command is available from the Geometry menu or from the right click shortcut menu.

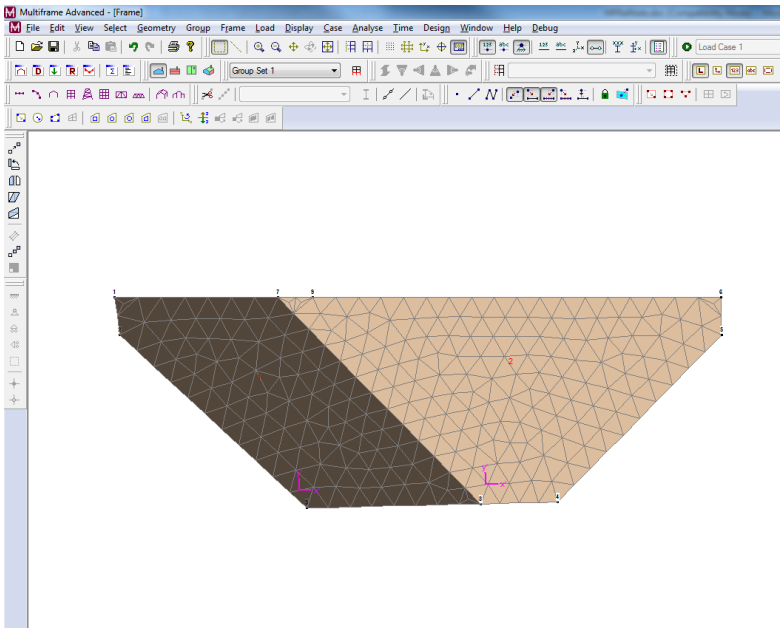
To split a patch

- **Choose Split Patch command**



- **Click on start & end point**



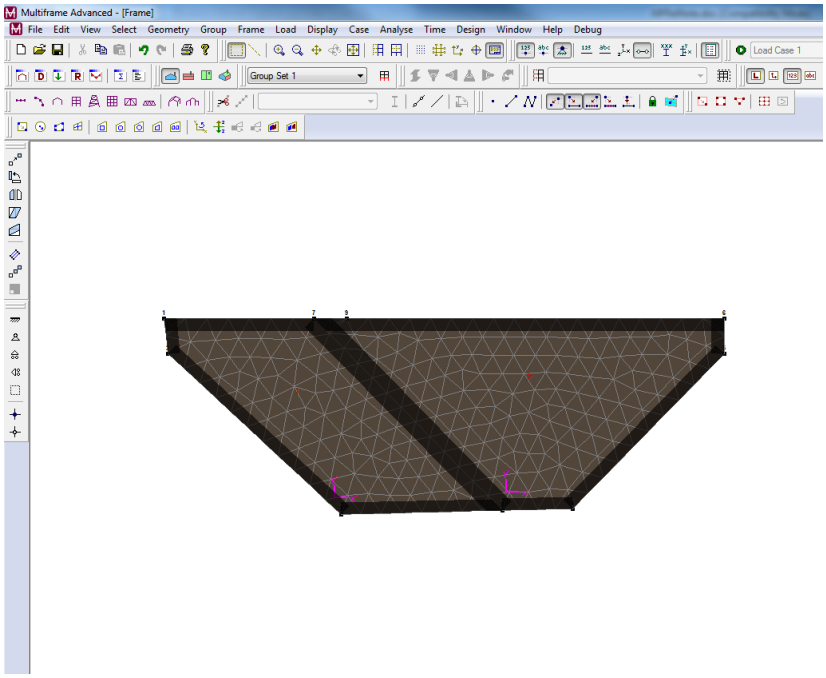


Patch Merge

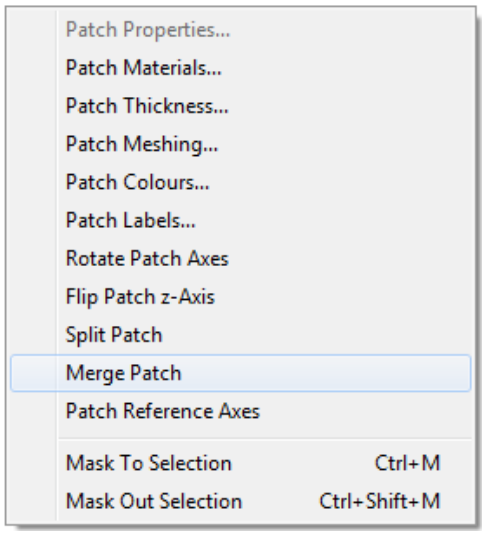
Multiframe allows patches to be merged into one planar patch. Patch merge command is available from the Geometry menu or from the right click shortcut menu.

To merge two patches

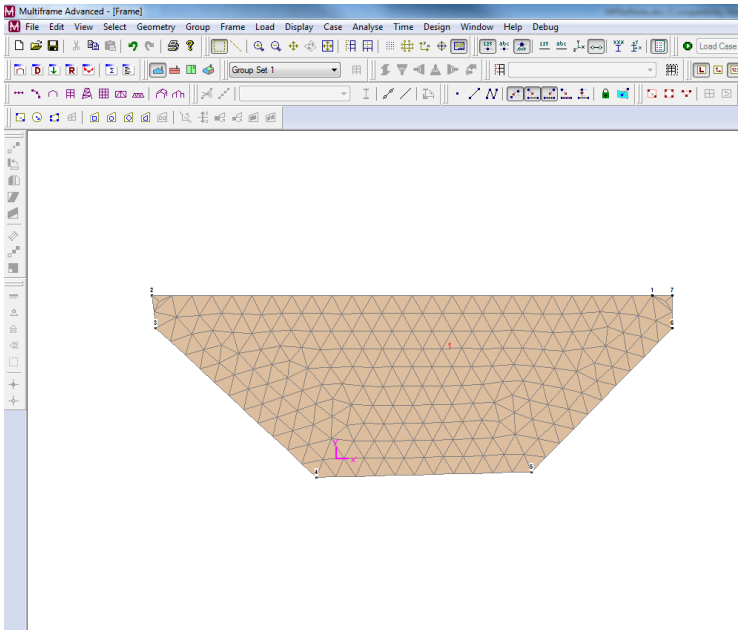
- **Select a list of adjacent patches in the same plane that will be merged**


Multiframe Advanced - [Frame]
File Edit View Select Geometry Group Frame Load Display Case Analyse Time Design Window Help Debug
Group Set 1
Load Case 1

- **Choose Merge Patch command**



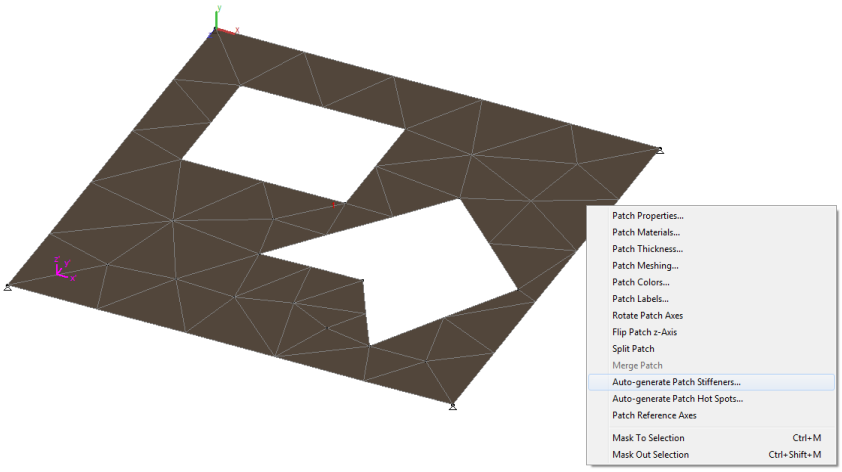
-



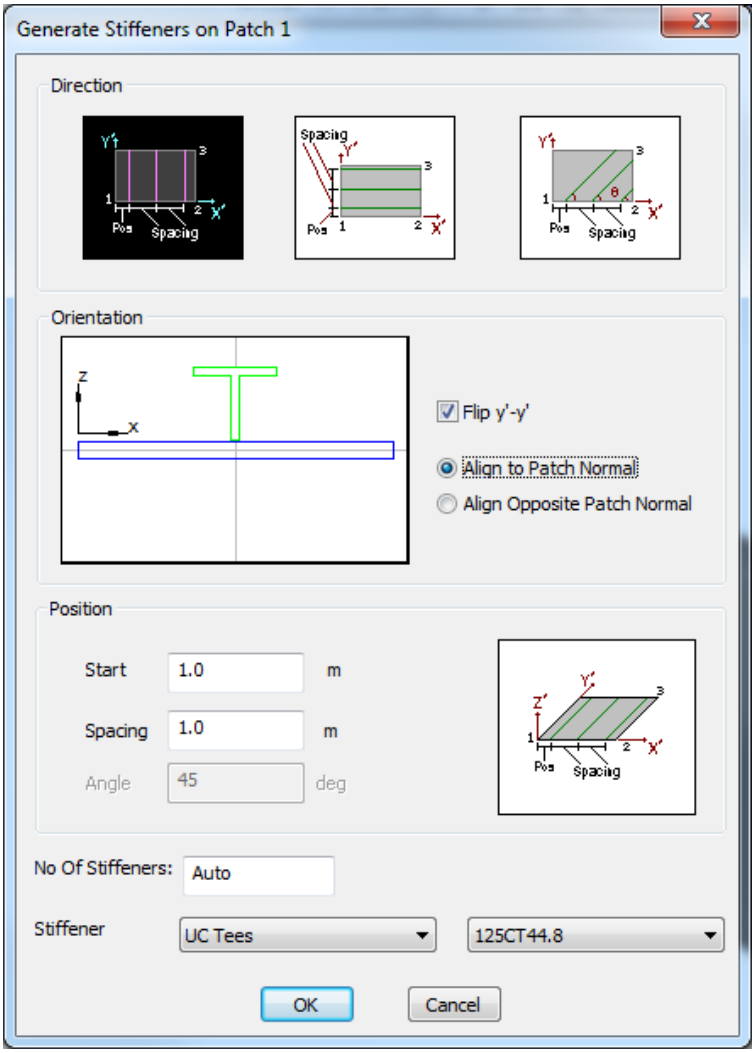
Auto-generate Patch Stiffeners

Multiframe supports patch stiffeners to be auto-generated. Patch stiffeners will be defined by its in-plane design members. Auto-generate Patch Stiffeners command is available from the Geometry menu or from the right click shortcut menu.

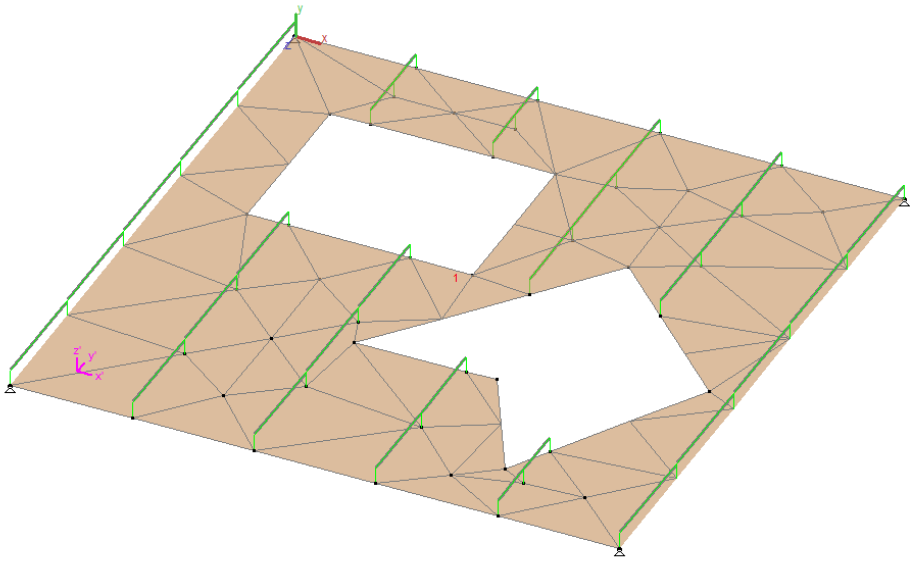
- **Select patch in which stiffeners will be generated in the Frame window, and mouse right click**



-
- **Choose Auto-generate Patch Stiffeners…**
- **A dialog box will appear with a field for the number of stiffeners, stiffener types, direction & section orientations etc.**



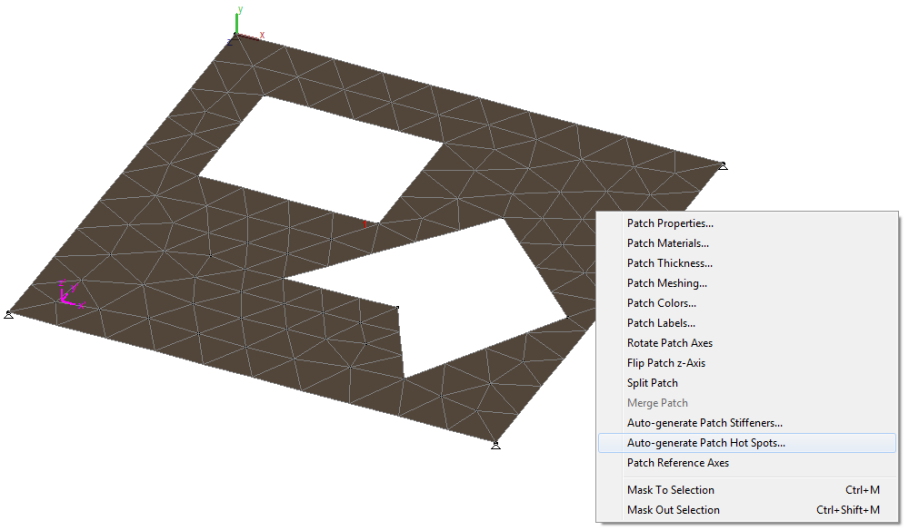
- **Click OK and will get patch stiffeners**



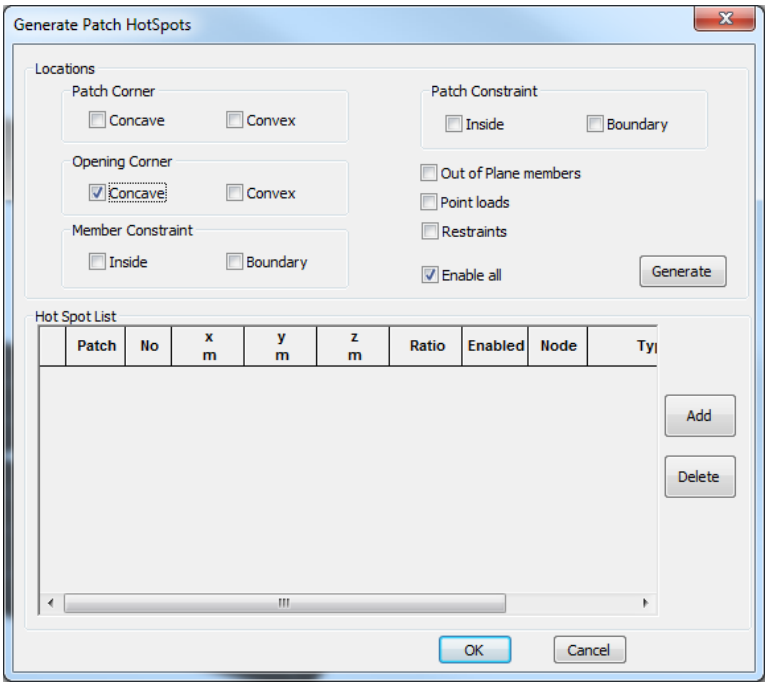
Auto-generate Patch Hot Spots

Multiframe supports patch hot spots to be auto-generated. Patch hot spots will be defined by patch corners, opening corners, constraint points and patch internal points. Auto-generate Patch Hot Spots command is available from the Geometry menu or from the right click shortcut menu.

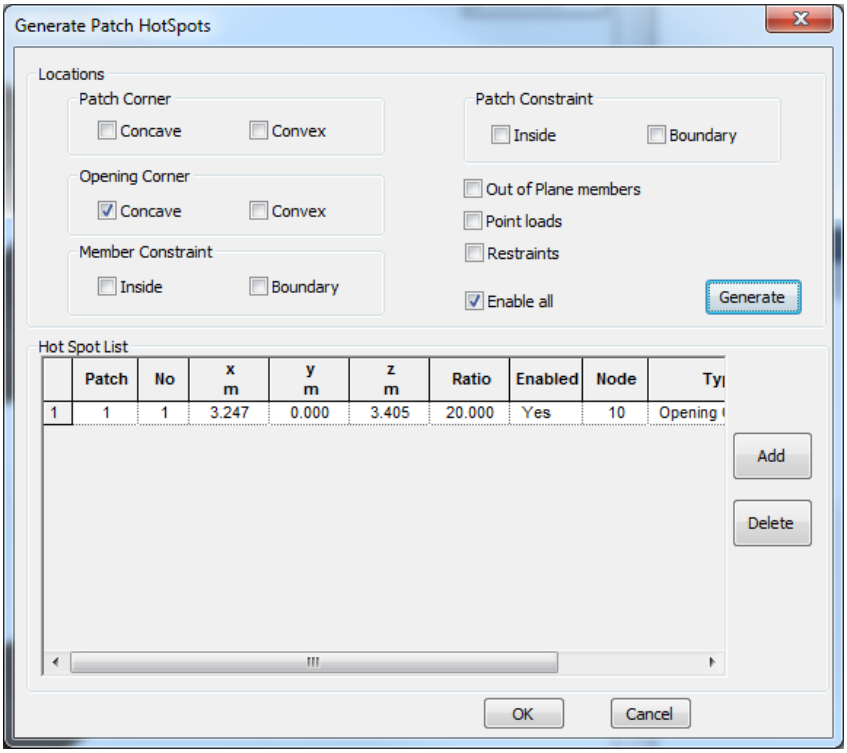
- **Select patch in which hot spots will be generated in the Frame window, and mouse right click**



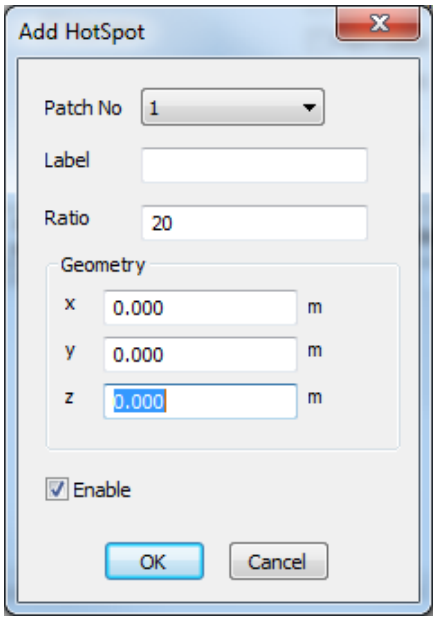
-
- **Choose Auto-generate Patch Hot Spots…**
- **A dialog box will appear with a field for type of hot spots, generate button & a list of hot spots etc.**



-
- **Select hot spot types, click generate button and will get hot spot list**

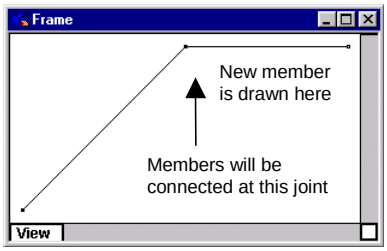


Multiframe allows uses to add or delete a hot spot from the above dialog. Click Add button, and will come up with a dialog window with fields for label, ratio, coordinates etc. Click OK button and hot spot will be added into the list.

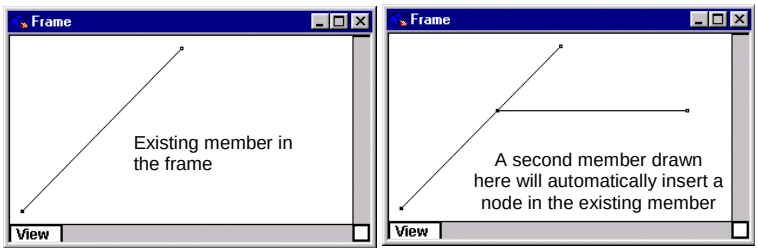


Connections Between Members

If the position of the first or second joint of the member you have drawn coincides with the position of an existing joint, the new member will be connected to the existing member at the existing joint. Note that this will happen even if the existing joint is at a different depth from the current drawing depth. For this reason, you will probably find it necessary to use the clipping and masking controls when drawing members which lie in front of or behind other members in the two dimensional views. When drawing in the 3D view, you can only draw a member between two existing joints.



The new member will also connect with an existing member if you start or finish drawing on the existing member.

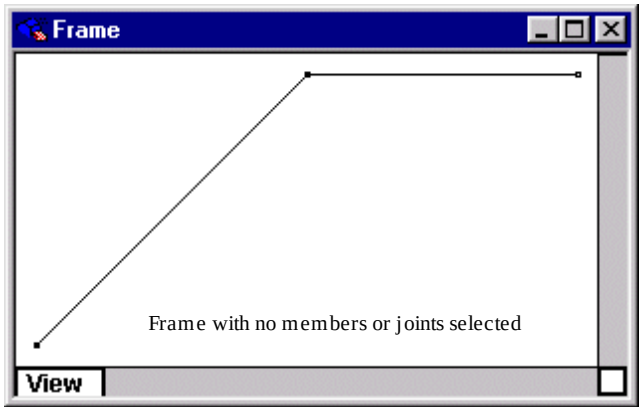


The connections between the members are set to be rigid. You can change this using the Member Releases command if you require a connection between members which cannot support moments.

Selections

In order to specify the properties, restraints and loads associated with joints and members, it is necessary to be able to identify which parts of the structure the various properties will be associated with.

You do this by graphically selecting joints and members prior to choosing menu commands. When you choose a command from a menu, to apply restraints at joints for example, the command will act on the joints that are currently selected. Note that you cannot select a joint or member that has been hidden using the clipping or masking commands.

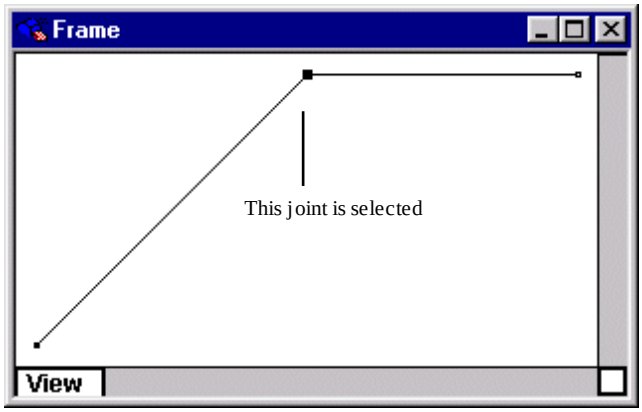


Selecting Joints

To select a joint

- **Click on the joint**

A selected joint is indicated by a solid black box around the joint.



To select a group of joints

- **Drag a rectangle which encloses the joints to be selected**

To select all the joints

- **Choose All from the Select menu**
- **Click the OK button**

To extend or reduce the selection

- **Shift-click to add a joint to or remove a joint from the current selection**
- **Shift-drag to add a group of joints to or remove a group of joints from the current selection**

If you know the number of the joint you wish to select, you can select it using the Joints command from the Select menu.

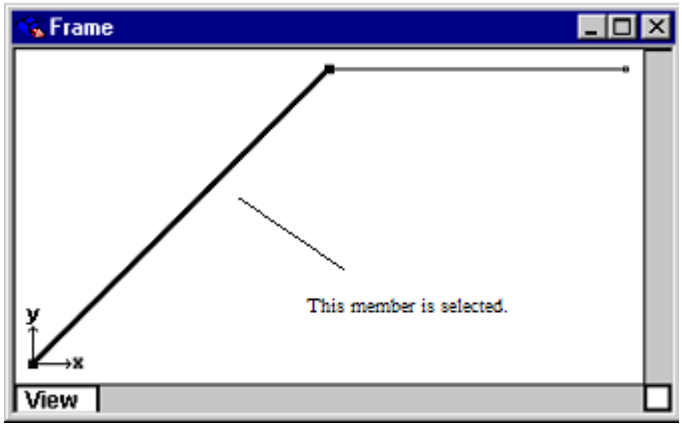
If more than one joint lies under the point where you click, Multiframe will select the joint closest to you.

Selecting Members

To select a member

- **Click on the member away from its ends**

A selected member is drawn with a heavy black line. A selected member always has the two joints at its ends selected.



If you know the number of the member you wish to select, you can select it using the Members command from the Select menu.

To select a group of members

- **Drag a rectangle which encloses the members to be selected**

If you drag from top-left to bottom-right, all of the members enclosed will be selected. If you drag from bottom-right to top-left, all of the enclosed members and all of the members intersecting with the rectangle will be selected.

To select all the members

- **Choose All from Select menu**

To extend or reduce the selection

- **Shift-click to add a member to or remove a member from the current selection.**
- **Shift-drag to add a group of members to or remove a group of members from the current selection**

When performing a drag selection there are two techniques for selecting a group of members and joints. The first is by using a rectangular box as described above.

An alternative to this is to use a line selection in which the user drags the end of a straight line across the screen. In this case all members that intersect this line are selected and well as all joints connected to these members.

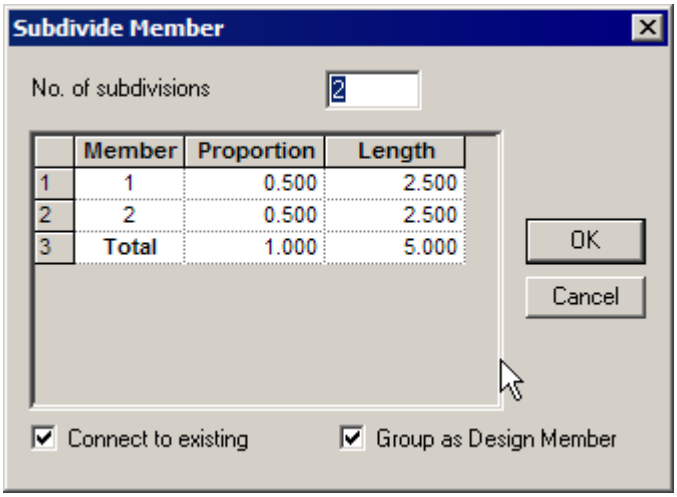
The selection tool to be used when performing a drag selection is picked using the View Toolbar. Clicking on the dotted square or dotted line buttons chooses the corresponding selection tool.

Subdividing a Member

To subdivide a member in the structure

- **Select the member or members to be subdivided**
- **Choose Subdivide Member from the Geometry menu**

A dialog box will appear with a field for the number of members you wish to subdivide the member into and a table listing the lengths of the new members.



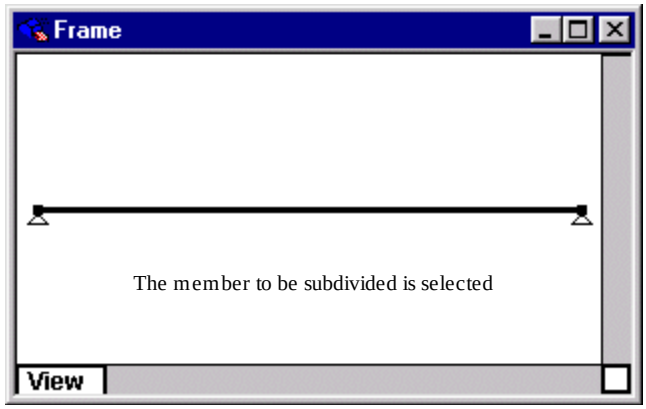
- **Enter the number of subdivisions to be created**

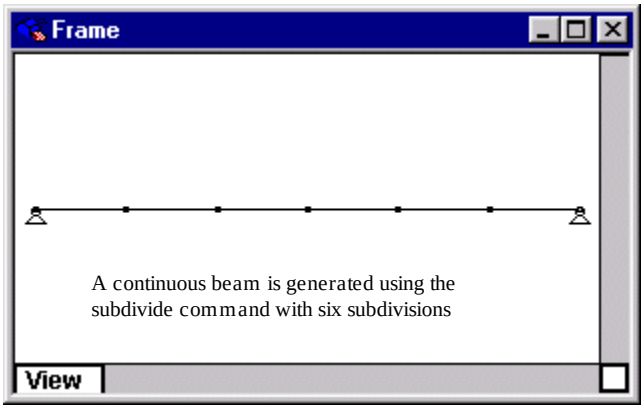
If you want the member to be subdivided evenly, leave the table of lengths unchanged otherwise, click in the table and change the length or proportion of the new members.

- **Select the "Group as Design Member" checkbox if the subdivided members are to be grouped as a design member.**
- **Click the OK button**

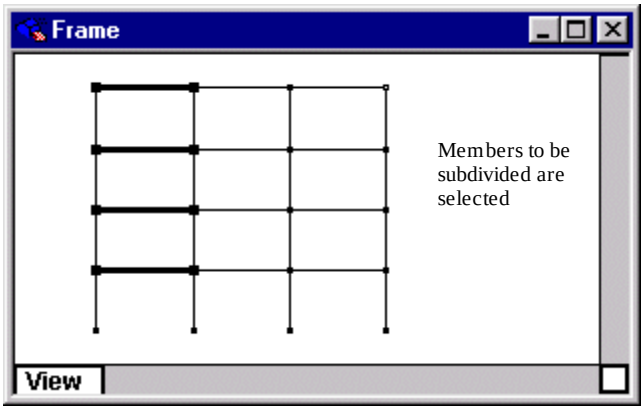
A number of new members connected end to end will be generated to replace each of the selected members. This command is particularly useful for generating continuous beams or for subdividing beams for the insertion of angle bracing.

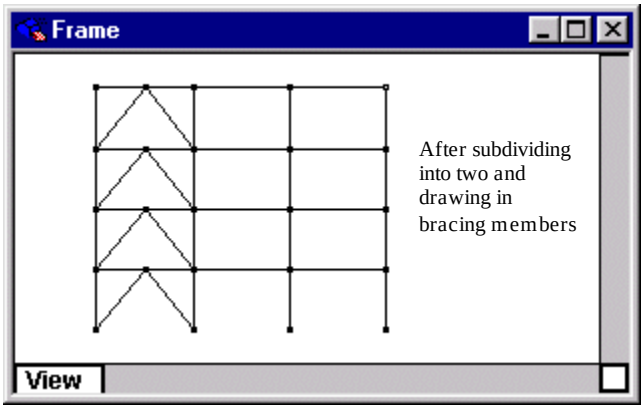
Generation of a continuous beam using the subdivide command.





Insertion of bracing in a frame by subdividing floor beams.



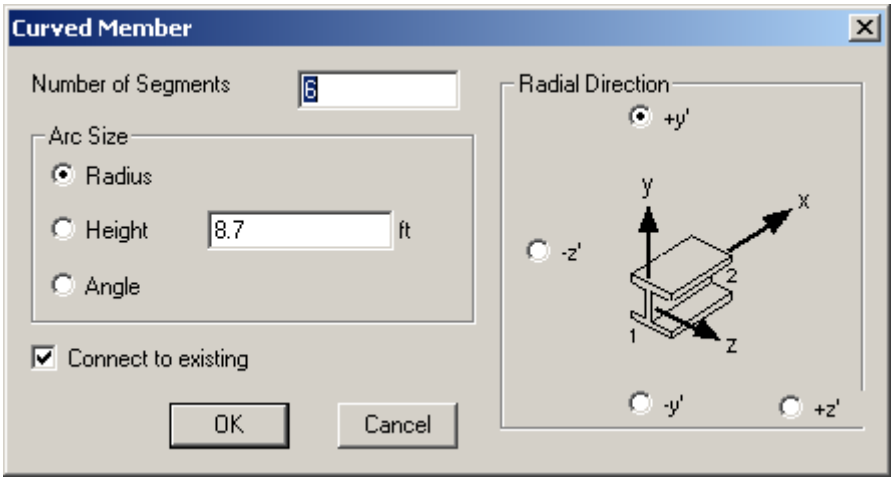


Converting a Member into an Arc

To convert a member in the structure into an arc

- **Select the member or members to be changed**
- **Choose Convert Member to Arc from the Geometry menu**

A dialog box will appear with a field for the number of members you wish to subdivide the member into and settings for the shape of the arc.



- **Enter the number of arc segments to be created**
- **Choose the size of the arc using one of the three methods**
- **Choose the direction of the arc relative to the member using the radio buttons on the right**
- **Click the OK button**

Normally, you will want the new members to connect with any existing members in the structure. If you do not want this to happen, uncheck the check box at the bottom of the dialog.

It is a good idea for you to test the accuracy of this approximation to a curve, by generating arcs with different numbers of members and examining the effect this has on the results of analysis.

Merge Members

The Merge Member command is used to automatically join members to create a single member. This command can be used to merge more than a single member at a time. Members to be joined into a single member are found by searching the current selection for all members that are of the same section type, a collinear, are rigidly connected and are off the same component type .To join members in a structure

- **Select the members to be merged**
- **Choose Merge Members from the Advanced submenu in the Geometry menu**

Groups of members that meet the above requirements will then be merged into a single member. In most cases the distributed loads applied to the merged member will also be merged to form continuous loads along the member. The user should carefully review the design properties of the merged members.

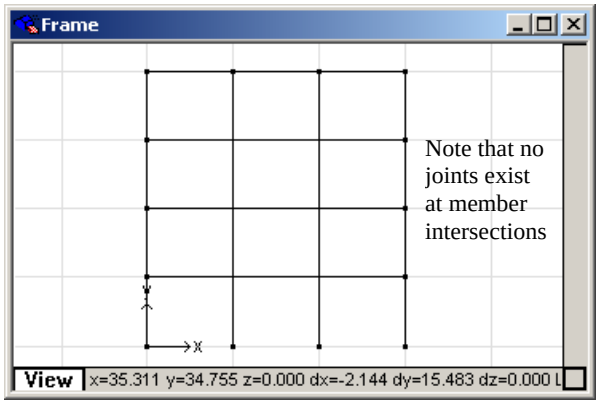
Intersect Members

The Intersect Member command is used to automatically subdivide members and connect members where they intersect. To join intersecting member in the structure

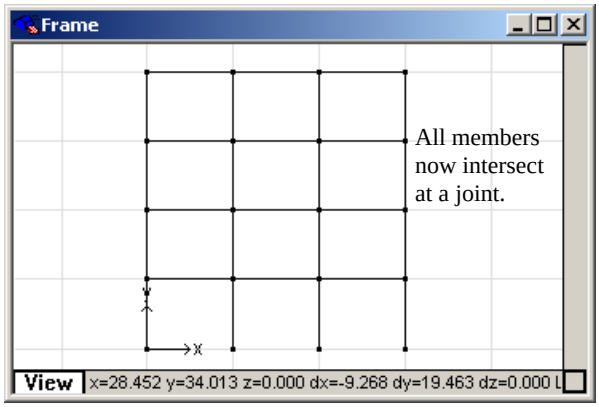
- **Select the members to be joined at there intersections**
- **Choose Intersect Members from the Advanced submenu in the Geometry menu**

A number of new members connected end to end will be generated to replace each of the selected members. This command is particularly useful for generating continuous beams or for subdividing beams for the insertion of angle bracing.

The intersection member command can be used to rapidly draw grillages and multistorey building. Continuous lines of beams or columns within the building can initially be drawn as a single member without regard for where they intersect.

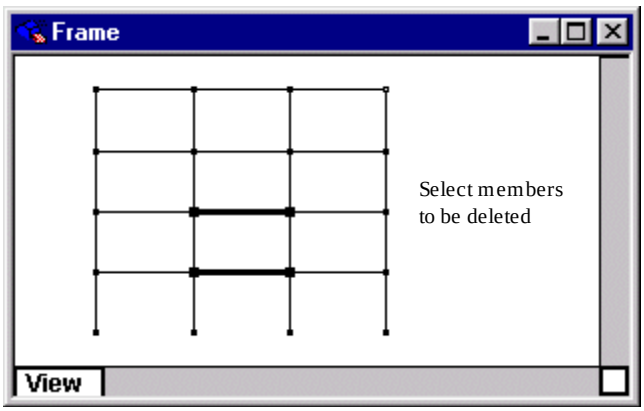


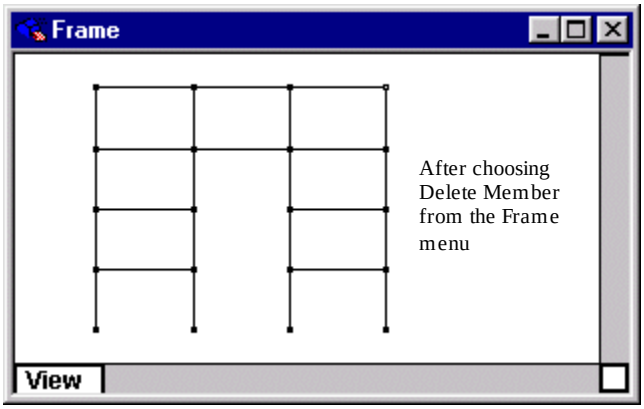
The Intersect Member command can then be used to correctly subdivide all the members in the model such that all intersecting members will be connected by a joint.



Deleting a Member

- **In the Frame window, select the member or members to be deleted**
- **Choose Delete Member from the Geometry menu or..**
- **Press the Delete key**





Automatic Generation

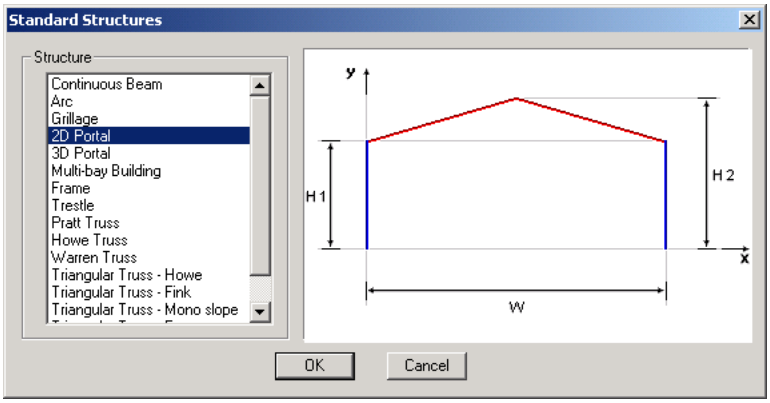
Multiframe includes capabilities for automatically generating commonly used frames. The four facilities provided are for standard bay and storied frames such as those found in multi-story buildings, portal frames as used in many buildings with sloping roofs, continuous beams and curved structures.

Generating a Portal Frame

To generate a portal frame

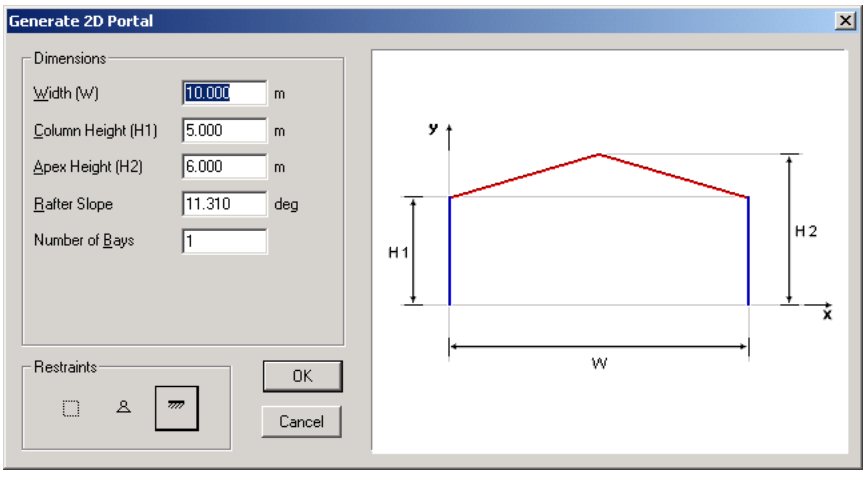
- **Choose Generate… from the Geometry menu**

A dialog box will appear with the a list of structures that can be generated.



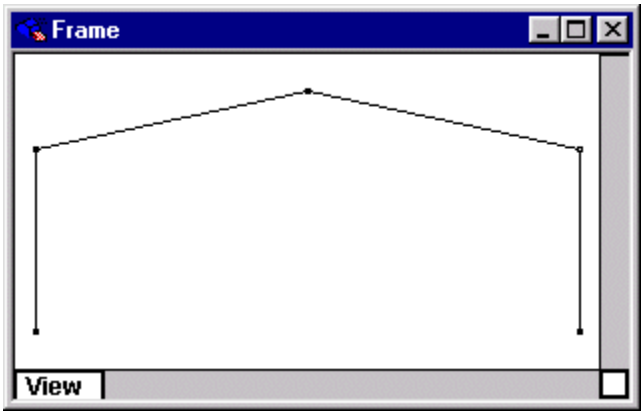
- **Select 2D portal item from the list and click OK**

A dialog box will appear allowing you to enter the dimensions of the frame.



- **Enter the values for the width, column height and the slope of the rafters**
- **Enter the values number of portal bays**
- **Select the type of restraint to be applied to the joints at the base of the columns**
- **Click the OK button**

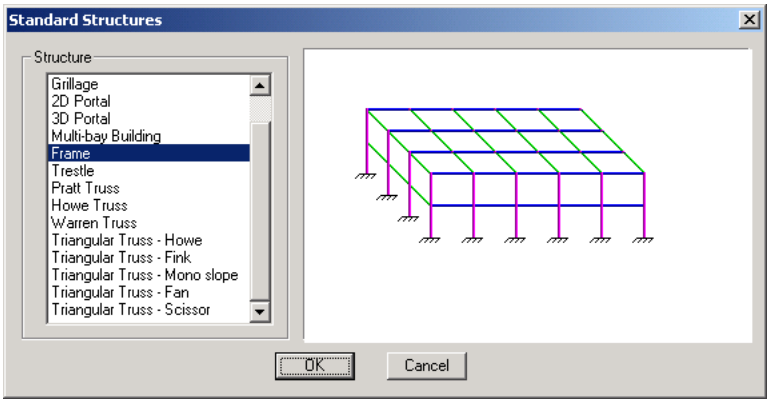
The pitched portal frame you generate will be shown in the Frame window.



Generating a Multi-story Frame

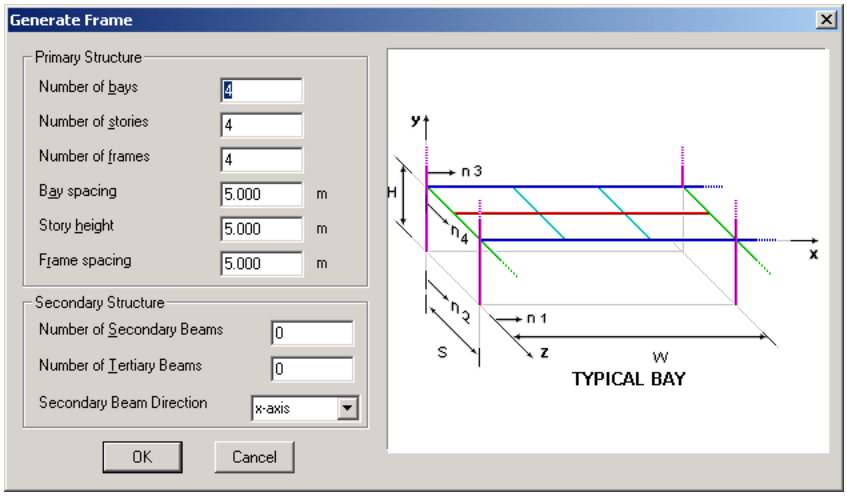
- **Choose Generate… from the Geometry menu**

A dialog box will appear with the various generation icons in it.



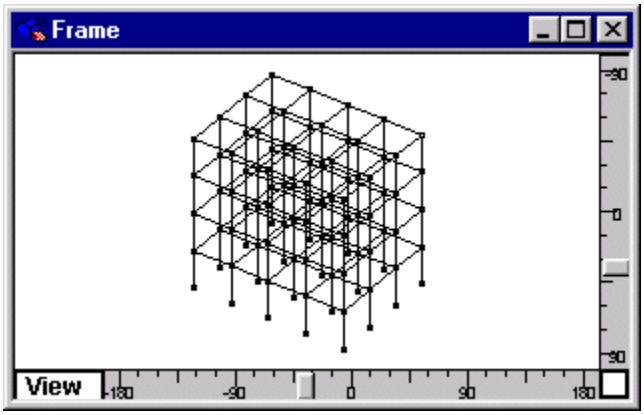
- **Click on the multi-bay frame icon and click OK**

A dialog box will appear with fields for the spacing in each direction. Bays run in the x direction, stories in the y direction and frames in the z direction.

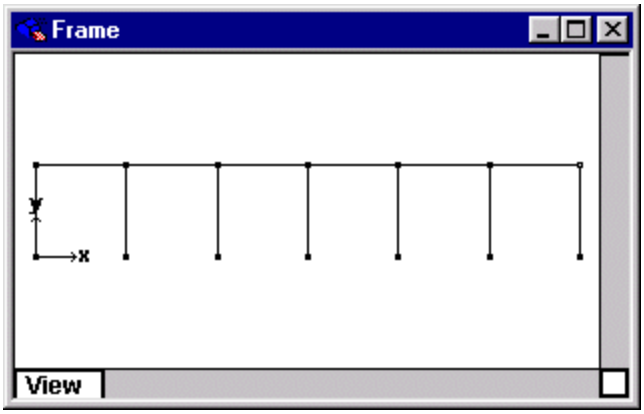


- **Enter the values for the number of structural elements and the dimensions in each direction.**
- **Enter the values for the number of secondary and tertiary beams in each bay of the frame.**
- **Select the direction of secondary beams**
- **Click the OK button**

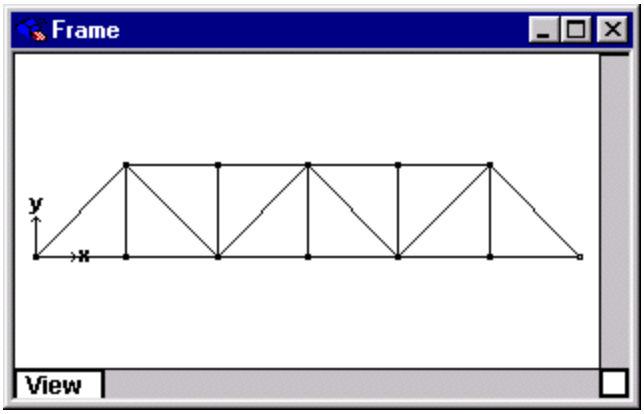
The frame you generate will be shown in the Frame window.



You may also find this command useful for generating the initial geometry for other structures such as trusses. You can start by generating the appropriate number of bays, 1 frame and 1 story.



You can then use the other drawing and duplication tools to create the structure you require e.g.



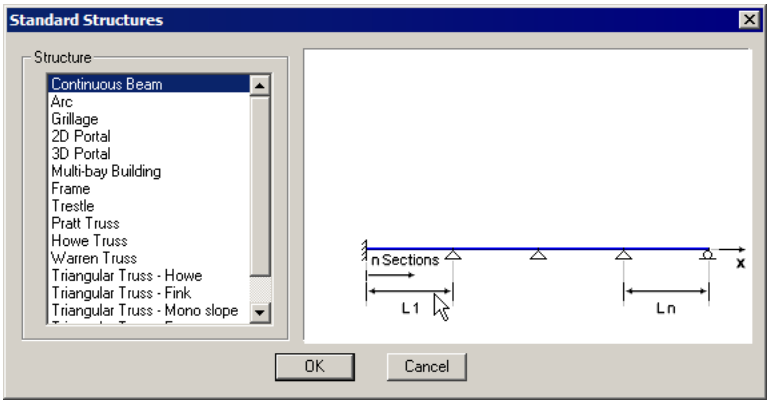
Generating a Continuous Beam

Multiframe allows you to quickly generate a continuous beam which is made up of a number of even or varying length spans.

To generate a continuous beam

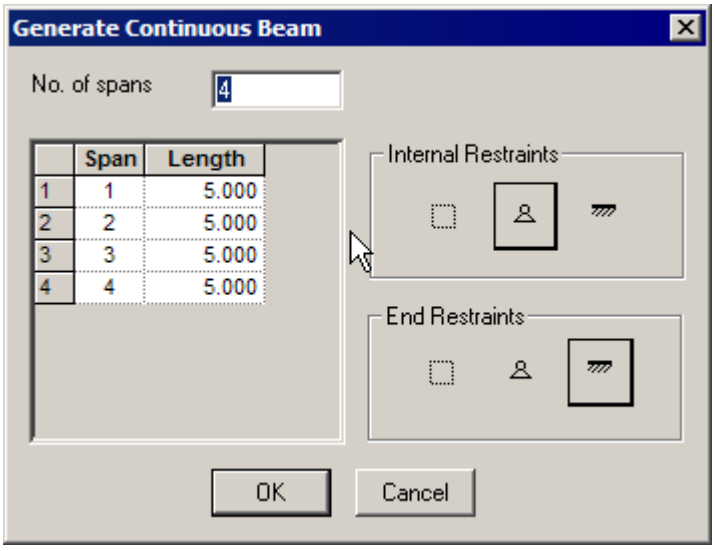
- **Choose Generate… from the Geometry menu**

A dialog box will appear with the various generation icons in it.

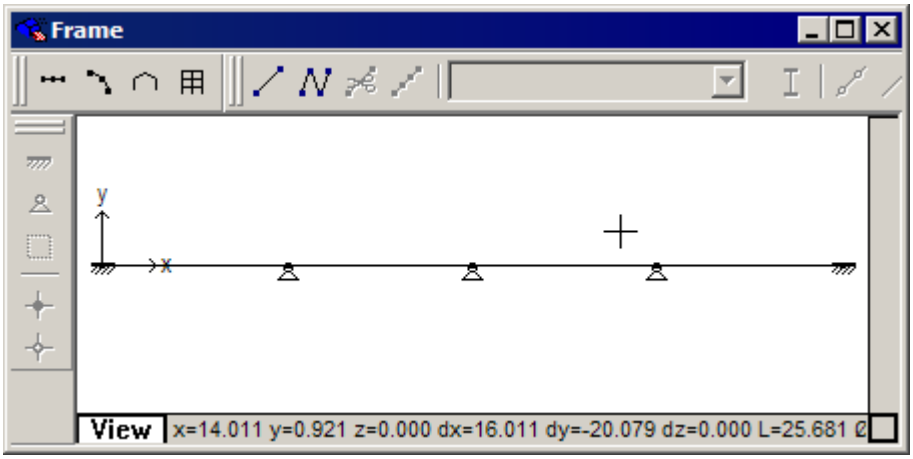


- **Click on the continuous beam icon and click OK**

A dialog box will appear with a table in it.



- **Type the number of spans in the beam**
- **Click on the first length value in the table to select it**
- **Enter the lengths for each of the spans using the Down Arrow key to move down the table**
- **Select the restraints to be applied to internal joints along the continuous beam.**
- **Select the restraints to be applied to the joints at the ends of continuous beam.**
- **Click the OK button**



Multiframe will automatically generate the beam with the dimensions you have specified.

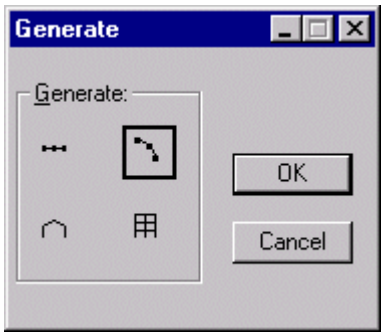
Generating a Curved Member

Multiframe allows you to quickly generate an approximation to a curved member by generating a number of short straight members.

To generate a curved member

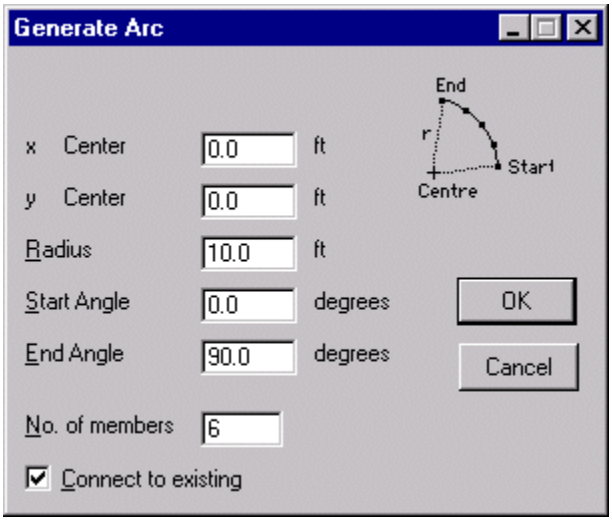
- **Choose Generate… from the Geometry menu**

A dialog box will appear with the various generation icons in it.

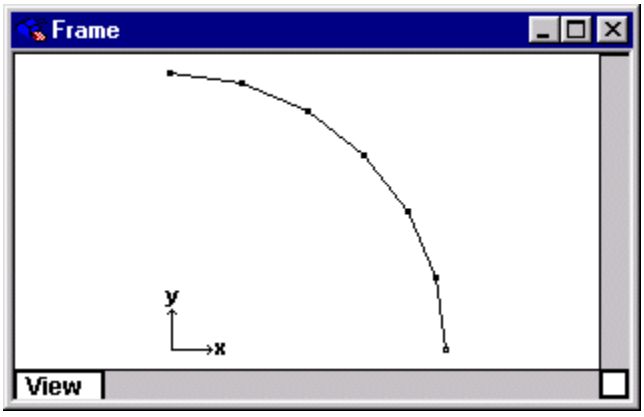


- **Click on the curved beam icon and click OK**

A dialog box will appear allowing you to enter the angles and radius of the beam.



- **Type the coordinates of the centre of the arc, radius and angle of sweep using the Tab key to move from number to number**
- **Type in the number of members you would like to use to approximate the curve**
- **Click the OK button**



Multiframe will automatically generate the assembly of members with the dimensions you have specified. The direction of the arc will depend on the current view in the Frame window. The arc will always be generated about the axis that is perpendicular to the screen or in the case of the 3D view, about the axis that is most perpendicular to the direction of view. This axis is not drawn in bold on the axis indicator.

Normally you will want the new members to connect with any existing members in the structure. If you do not want this to happen, uncheck the check box at the bottom of the dialog.

It is probably a good idea for you to test the accuracy of this approximation to a curve, by generating arcs with different numbers of members and examining the effect this has on the results of analysis.

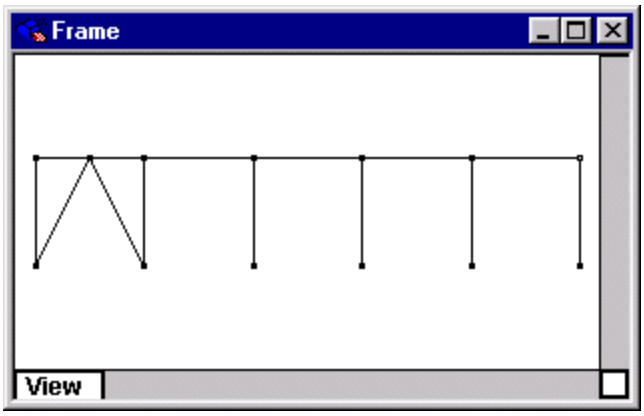
Generating a Regular Frame

Multiframe allows you to quickly generate a regular frame that is made up of a number of evenly spaced, similarly shaped sub-structures. Typical examples of this would be high-rise buildings, trusses or multiple bay portal frames. Multiframe also allows you to duplicate shapes in cylindrical and spherical coordinates.

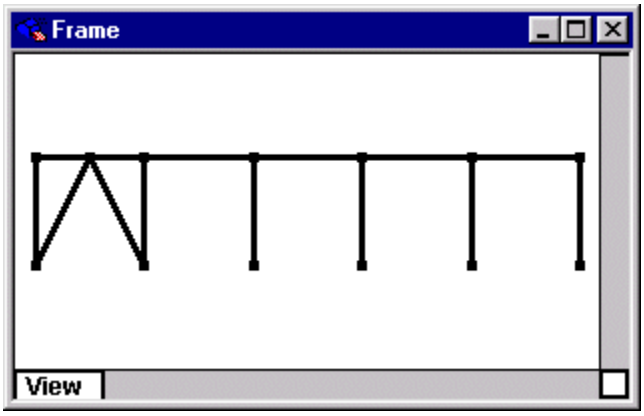
Duplicate

To duplicate a structure

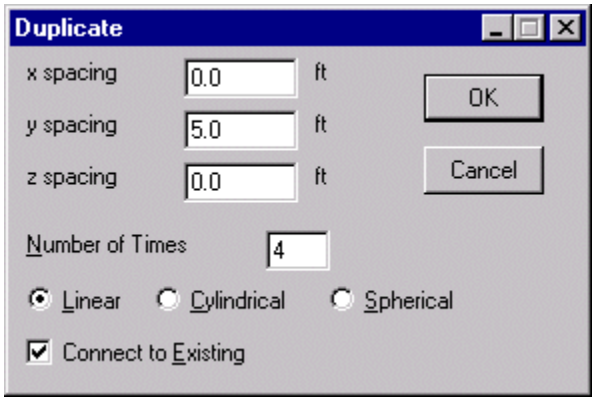
- **Draw or generate the sub-structure you wish to duplicate**



- **Select the sub-structure to be duplicated**

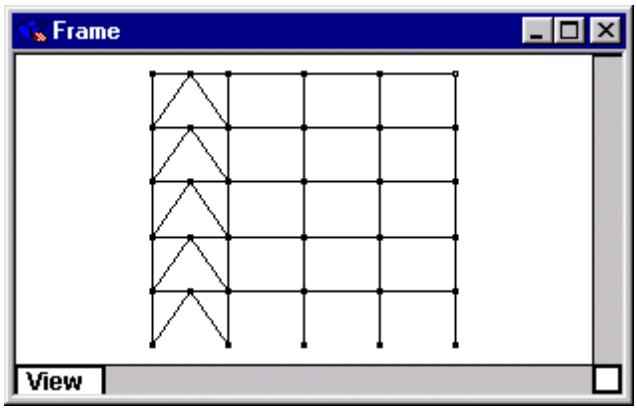


- **Choose Duplicate… from the Geometry menu**



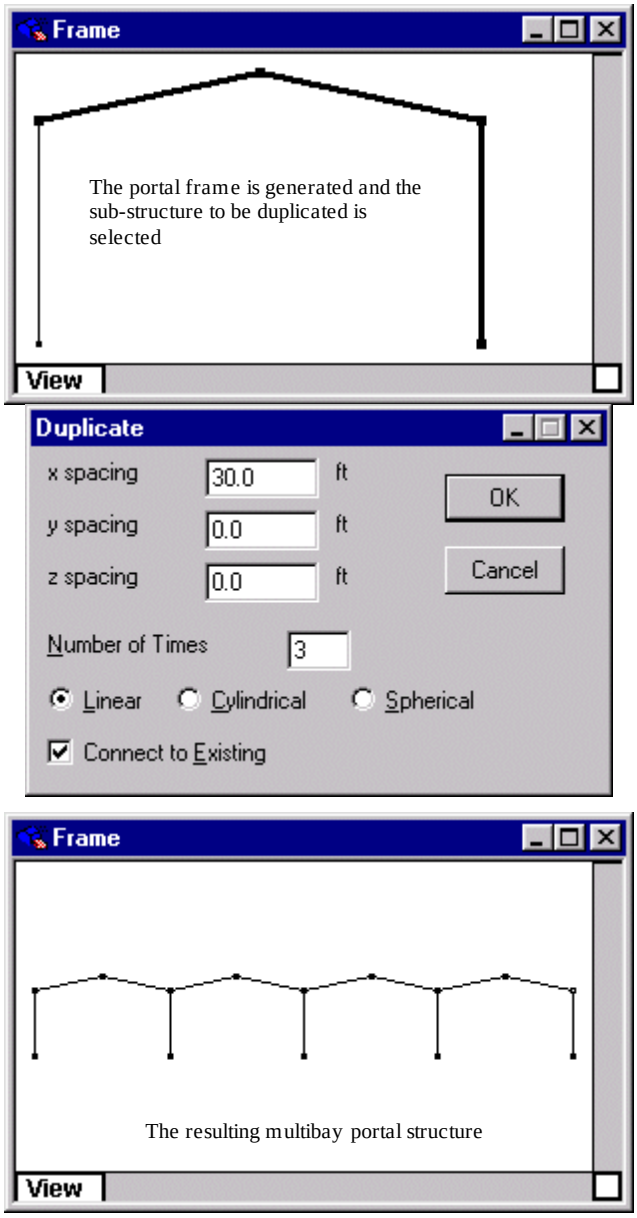
A dialog box will appear allowing you to specify the type of duplication, the spacing in each direction, the number of times the selection is to be duplicated and whether the duplicated members should be connected with the existing structure.

- **Choose which type of duplication you require**
- **Enter the spacing for your sub-structure**
- **Enter the number of times you wish to repeat the sub-structure**
- **Click the OK Button**



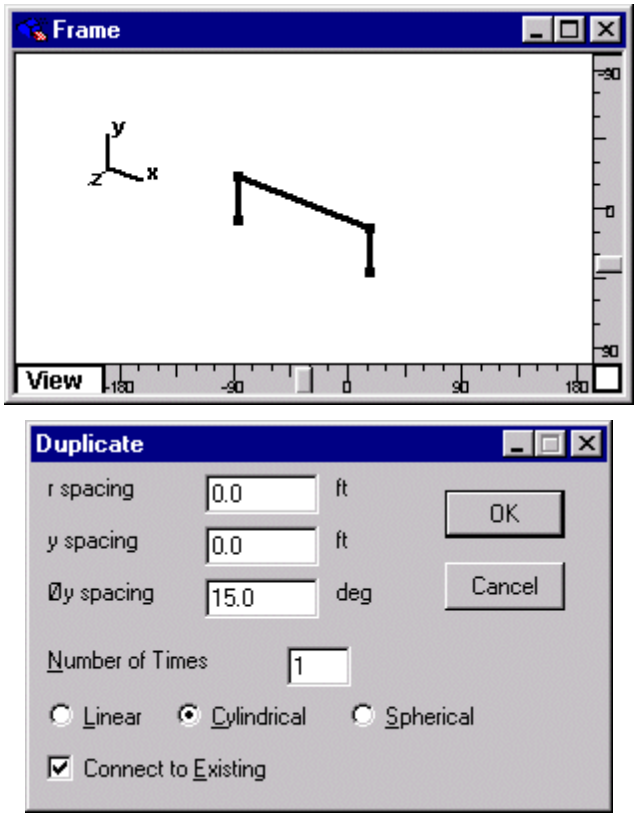
If you do not want connections to be generated between adjacent members, then switch off the check box ‘connect to existing’. Normally you will want this option left on. A connection will be made between generated members and existing members if the end joints are within 0.2 inches (5 mm) of each other.

An example of the use of the Duplicate command to generate a multiple bay portal frame is shown below.

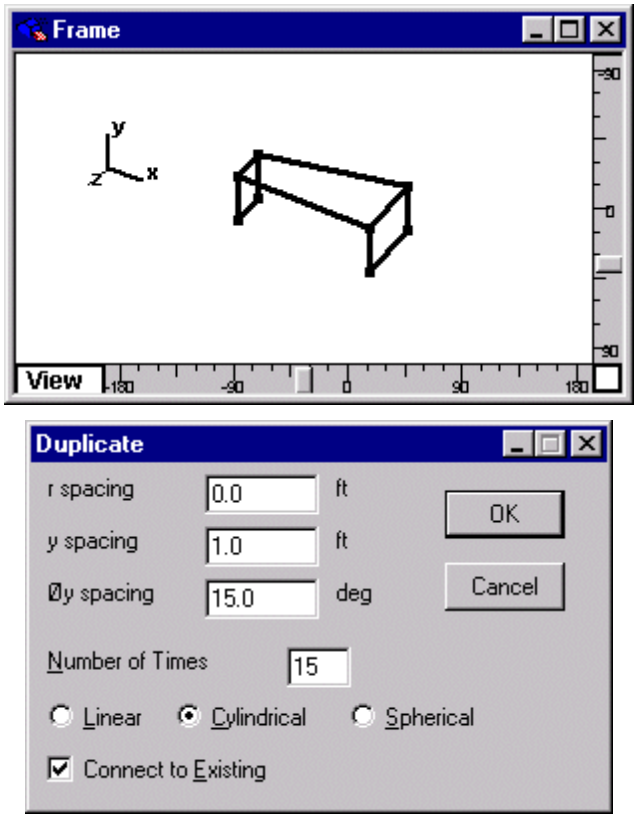


Cylindrical Coordinates

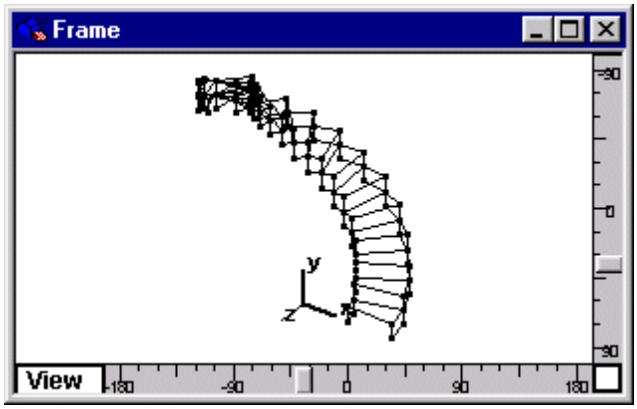
If you wish to generate a structure in cylindrical coordinates, you can select the cylindrical option. This then allows you to space the duplication radially from the y axis, angularly about the y axis, or linearly along the y axis. As an example, to generate a circular staircase you would first draw half of one step and then duplicate it to form the other half.



This would form one step of the staircase.



Then you would select the whole step and duplicate it to form the entire staircase.



Spherical Coordinates

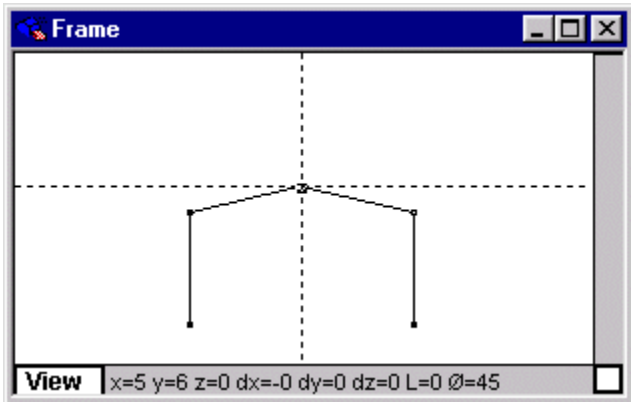
If you wish to generate a structure in spherical coordinates, you can select the spherical option. This then allows you to space the duplication radially from the origin, angularly about the z axis, and angularly about the y axis. As an example, to generate a cylindrical vault you could first draw a continuous beam parallel to the z axis and then duplicate it angularly about z to form the vault.

Moving a Joint

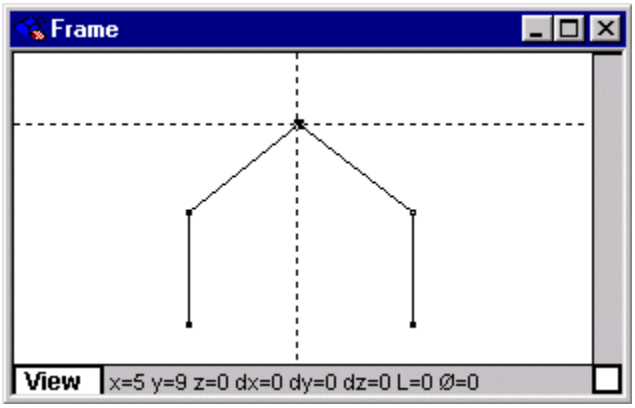
You can only move joints in the Frame window. Before moving a joint, be sure that there are no joints selected in the frame.

To move a joint

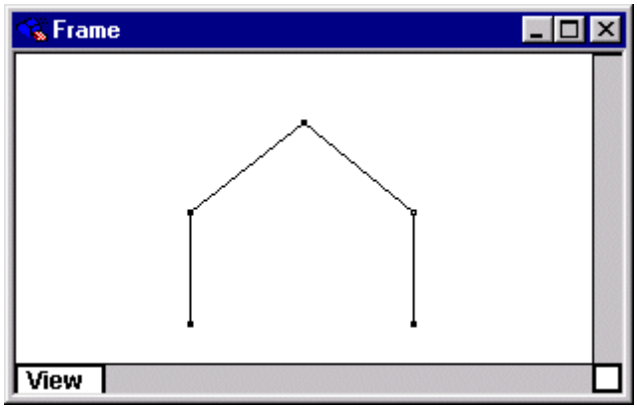
- **Point to the joint, press the mouse button and hold it down**



- **Drag to the new position of the joint**



- **Release the mouse button to fix the new location**



Holding down the shift key while dragging will constrain movement vertically, horizontally or to 45 degrees. The coordinates of the joint will be displayed in the bottom left hand corner of the window as you drag. If you have turned on the Grid option, the joint will align with the grid as you move it.

If you drag a joint on top of an existing joint, Multiframe will create a connection between the members that meet at the common point.

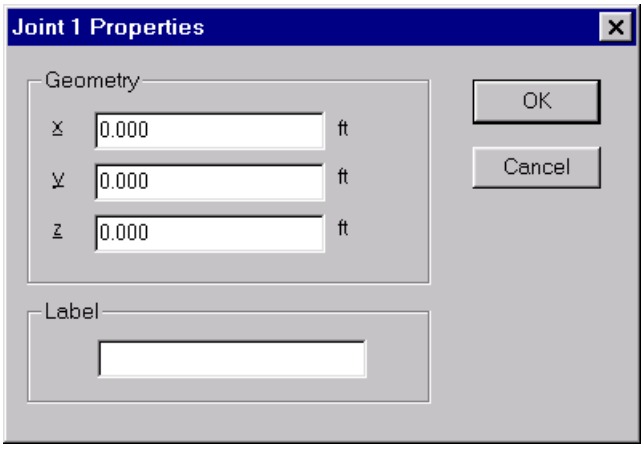
You can only drag joints in two-dimensional views. When you drag a joint, Multiframe will not change the depth of the joint.

Typing Joint Coordinates

You can also move a joint by typing in new coordinates for its position.

- **Double click on the joint**

A dialog box will appear with the joint's coordinates.



- **Type in the new values for the coordinates**
- **Click the OK button**

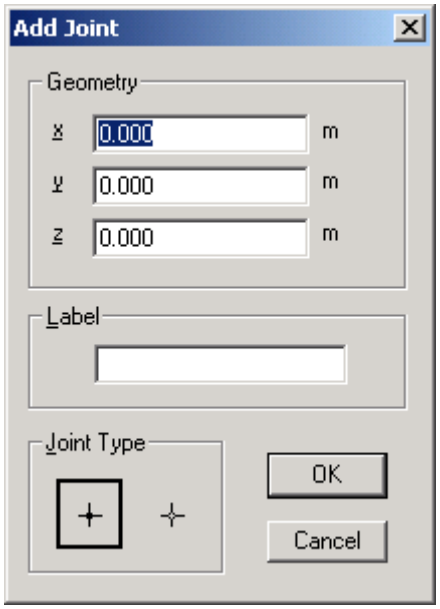
You can also change a joint's location by typing or pasting new values in the X, Y and Z columns of the Joint table in the Data window.

Adding Joints

As joints are automatically added to the model when members are drawn there is generally no need to add individual joints to a frame. However, in some circumstances it can be more convenient to add joints at specified locations and later draw in the members connected to the joint. To add a new joint to the model

- **Choose Add Joint… from the Advanced submenu in the Geometry menu**

A dialog box will appear prompting for the coordinates and properties of the new joint.



- **Type in the values for the coordinates**
- **Optionally enter an label for the joint**
- **Select the joint type**
- **Click the OK button**

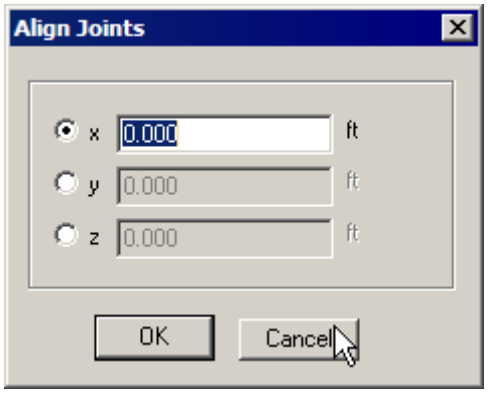
Upon successful exit from the dialog a new joint will be added to the frame at the specified location.

Aligning Joints

Joints in the model can have a single component of their position set to a specified value so as to align the joint to a specific location. To align a number of joints to the same position

- **Select the joints to be aligned in the Frame window**
- **Choose Align Joints… from the Advanced submenu in the Geometry menu**

A dialog box will appear prompting for the axis and position along that axis to which all the selected joints will be reposition to.



- **Select axis to which along which the joints will be aligned**
- **Type in the values for the new coordinate**
- **Click the OK button**

Upon successful exit from the dialog all the selected joints will be have the specified coordinate set to the new value.

Moving a Group of Joints

Multiframe also allows you to move a group of joints together either by dragging with the mouse or by typing in a distance to move.

To move a group of joints with the mouse

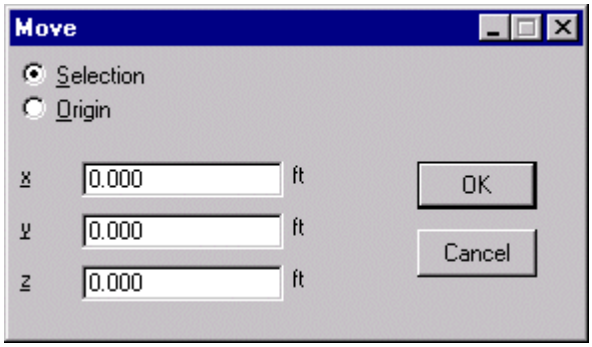
- **Select the joints to be moved**
- **Point to any of the selected joints and press the mouse button**
- **Drag the joints to the new location**
- **Release the mouse button**

Multiframe will connect together any members that share a common end point after you have dragged the joints. Holding down the shift key while dragging will constrain movement vertically, horizontally or to 45 degrees. The coordinates of the pointer will be displayed in the bottom left hand corner of the window as you drag. If you have turned on the Grid option, the joints will move in increments of the grid spacing as you move.

To move a group of joints numerically

- **Select the joints to be moved**
- **Choose Move… from the Geometry menu**

A dialog box will appear with fields for the distance you wish to move the joints in each direction. The Selection radio button will be selected indicating that you wish to move the selected joints rather than the origin.



- **Enter the distance the joints are to be moved in each direction**
- **Click on the OK button**

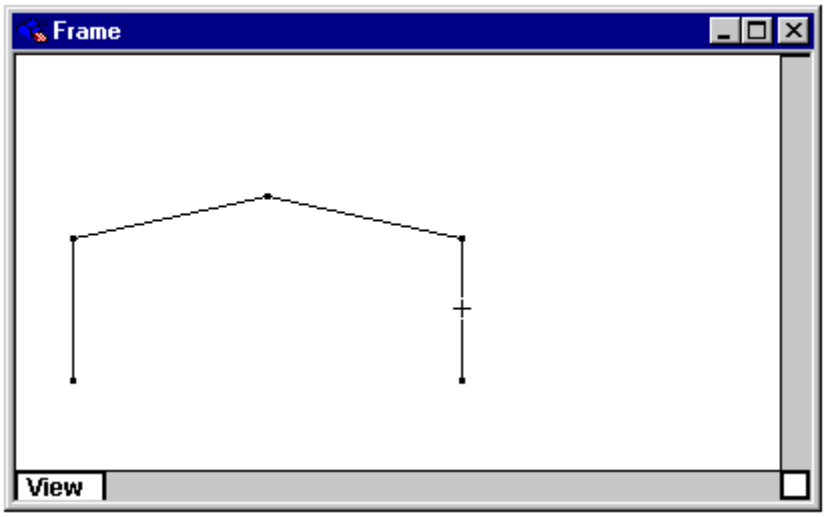
Multiframe will create a connection between joints that you superimpose on top of each other using this command. You can also move a single joint using this command. Simply select one joint before using the command.

Moving a Member

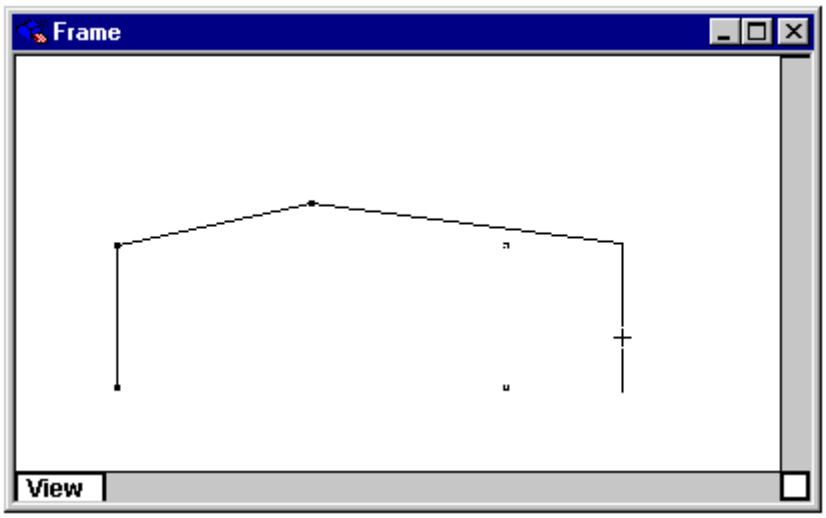
You can only move members in the Frame window. Before moving a member, make sure that there are no members selected in the frame.

To move a member

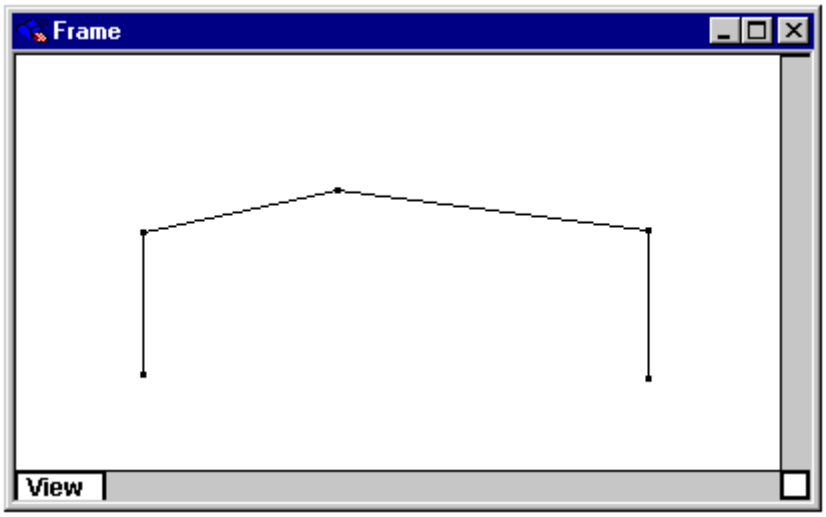
- **Point to the member away from its ends and press the mouse button**



- **Drag the member to its new position**



- **Release the mouse button to fix the new position of the member**



The slope and length of the member will remain constant as you move the member. You can only move members in the two dimensional views.

Holding down the shift key while dragging will constrain movement vertically, horizontally or to 45 degrees. The coordinates of the pointer will be displayed in the bottom left hand corner of the window as you drag. If you have turned on the Grid option, the member will move in increments of the grid spacing as you move.

If you drag a joint on top of an existing joint, Multiframe will create a connection between the members that meet at the common point.

Disconnecting Members

When moving part of a frame it is sometimes necessary to disconnect that part of the frame from the rest of the structure. To disconnect members from a model

- **Select the members to be disconnected from the rest of the frame**
- **Choose Disconnect Members from the Advanced submenu in the Geometry menu**

The command creates new joints at all the joints that connect the selection to the rest of the model. These new joints are then used to define the topology of the disconnected parts of the frame.

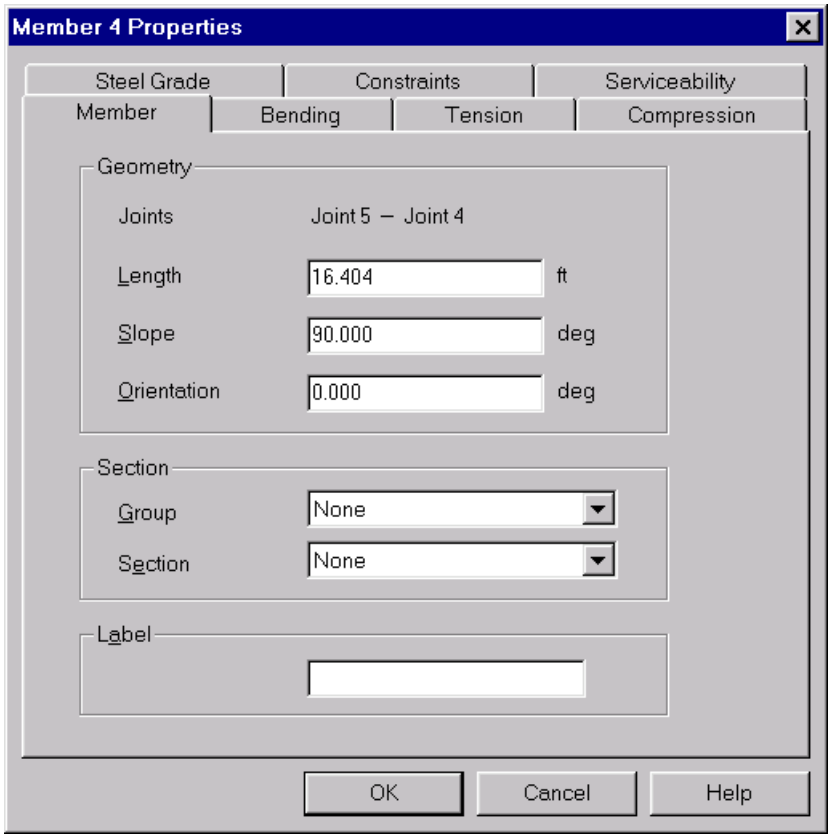
Resizing a Member

You can change the length or slope of a member in the Frame window by typing in new values.

To change the length or slope of a member

- **Double click on the member**

A dialog box will appear with the members name, end joint numbers, section type, length, slope and orientation.



- **Type in new values for the properties you wish to change**
- **Click the OK button**

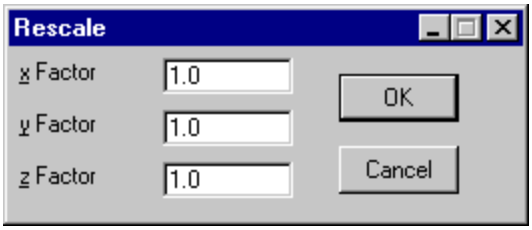
Joint 1 will stay at its original location, joint 2 will be moved to match the entered length and slope.

Rescaling the Structure

You can use the Rescale… command from the Geometry menu to rescale the selected joints in the Frame window. This command is useful for investigating structural alternatives where you may wish to change the overall aspect ratio of a structure or part of a structure.

- **Select the part of the structure to be rescaled**
- **Choose Rescale… from the Geometry menu**

A dialog box will appear with the scaling factors in each of the axis directions.



- **Enter the scaling factors for the x, y and z directions**
- **Click the OK button**

Rescaling is done relative to the origin of the axes, so you may wish to use the Move… command to move the origin prior to rescaling, to ensure that rescaling occurs from the correct point. Coordinates are rescaled by multiplying their value by the scaling factor. For example, a joint with coordinates x=2, y=2, z=2 and rescaled using scale factors of 1.5, 2.0 and 3.0 for the x, y and z directions respectively, would move to the new location of x=3, y=4, z=6.

Using a scaling factor of -1 will reflect the selected joints about the appropriate axis.

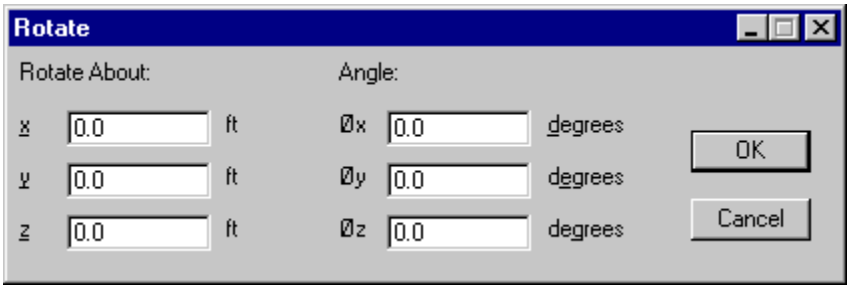
Multiframe will not create a connection between joints that you superimpose on top of each other using this command.

Rotating Members

You can use the Rotate… command from the Geometry menu to rotate the selected joints in the Frame window.

- **Select the part of the structure to be rotated**
- **Choose Rotate… from the Geometry menu**

A dialog box will appear with the rotation centres and angles.



- **Enter the coordinates of the axis you wish to rotate the selection about**
- **Enter the magnitude of the rotation for the appropriate axis**
- **Click the OK button**

Rotating is done relative to the axis you specify with the coordinates. For example, suppose you wanted to rotate the whole frame by 30° about a line parallel to the z axis passing through the point x=10, y=20. You would first select the whole frame, and then you would enter x=10, y=20, z=0, Øx=0, Øy=0, Øz=30 in the Rotate dialog.

Multiframe will not create a connection between joints that you superimpose on top of each other using this command.

When rotating the selected joints you should enter only one non-zero angle. That is, Multiframe can only do one rotation at a time.

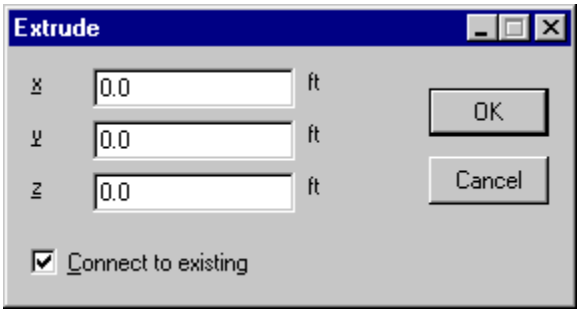
Extruding Beams or Columns

When you wish to add beams or columns to an existing frame, you may find it easier to use the Extrude command rather than drawing the new members with the mouse. The Extrude command allows you to project a member or group of members in any of the global axis directions from existing joints. For example, you could draw a floor plan and then extrude the columns up from it or you could extrude the beams out from a wall frame you have drawn.

To extrude members from the frame

- **Select the joints in the frame to be extruded**
- **Choose Extrude… from the Geometry menu**

A dialog box will appear with fields for the direction and dimension of the extruded members.



- **Enter the length of the extruded members in the appropriate direction**
- **Click the OK button**

If you leave the Connect to existing check box checked, Multiframe will automatically create a connection between the selected joints and the extruded members. It will also connect the extruded members to any existing members in the frame if they lie within a 0.2 in (5mm) tolerance of the ends of the existing members.

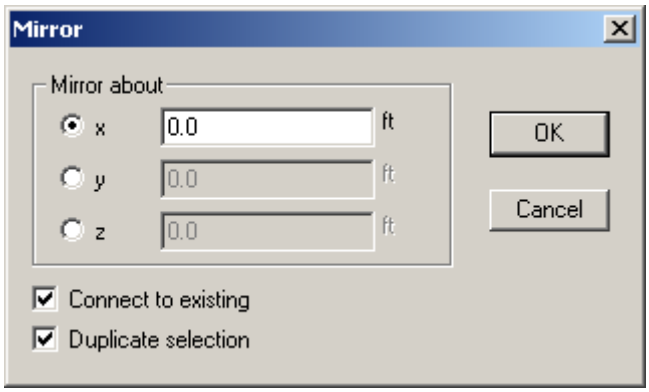
Mirroring Members

When you wish to create a structure with one or more axes of symmetry, you can use the Mirror command to mirror the structure you have drawn.

To mirror members in a frame

- **Select the members to be mirrored**
- **Choose Mirror… from the Geometry menu**

A dialog box will appear with fields for the direction to mirror and the location of the plane about which mirroring will take place.



- **Choose the radio button of the plane about which mirroring will occur**
- **Enter the location of the plane along that axis**
- **Click the OK button**

If you leave the Connect to existing check box checked, Multiframe will automatically create a connection between the selected joints and the mirrored members.
If you leave the Duplicate Selection check box checked, Multiframe will duplicate the selected members otherwise the existing members will be moved.

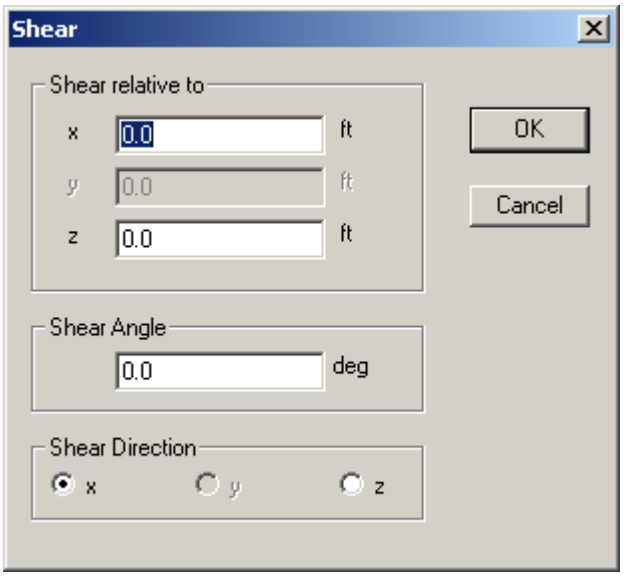
Shearing Members

When you wish to create a structure which has a sheared shape, such as a bridge deck, you can use the Shear command to transform and rectangular structure into a sheared shape.

To shear a group of members in a frame

- **Select the members to be sheared**
- **Choose Shear… from the Geometry menu**

A dialog box will appear with fields for the origin from which the shear angle will be measured, the direction to shear and the angle of the shear.



- **Click OK to shear the members**

Editing Coordinates Numerically

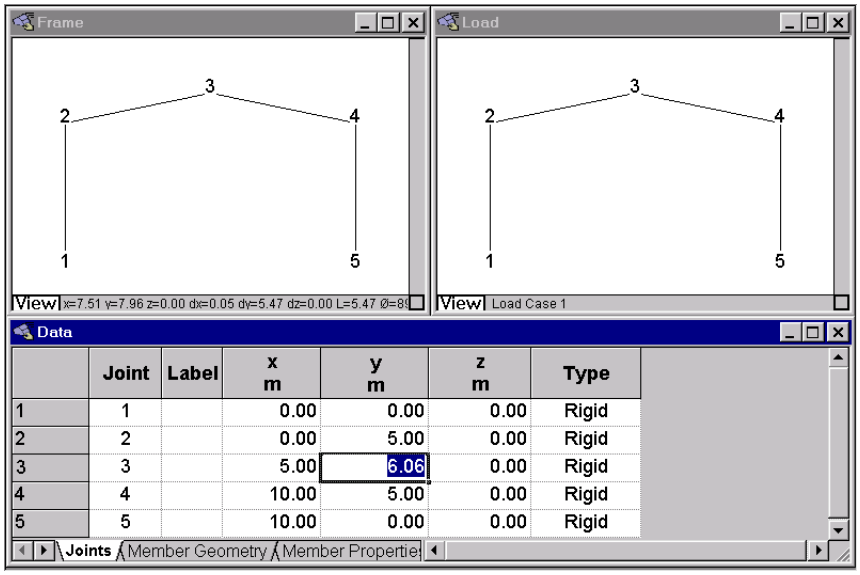
Multiframe allows you to change the positions of joints in the structure numerically rather than graphically. You will find this option useful for "fine tuning" the coordinates of your structure after drawing it.

Each joint in the structure is identified by a unique number. You can display these numbers in the Frame, Load and Plot windows by choosing Symbols… from the Display menu, checking the Joint Numbers check box and clicking the OK button.

Coordinates are changed by typing values in the Data window or by double clicking on the joint in the Frame window. You will probably find it helpful to use the Editing Layout from the Window menu while you are typing in coordinates, as this will allow you to see the changes to the structure as you make them. The coordinates are displayed in a table in the Data window. You can resize the columns in the table by dragging the lines that separate the columns. You can change the text font and size used in the table by using the Font… command from the View Menu while the Data window is in front on the screen. You can control the format of the numbers displayed in the table by using the Numbers… command from the Format window while the Data window is in front.

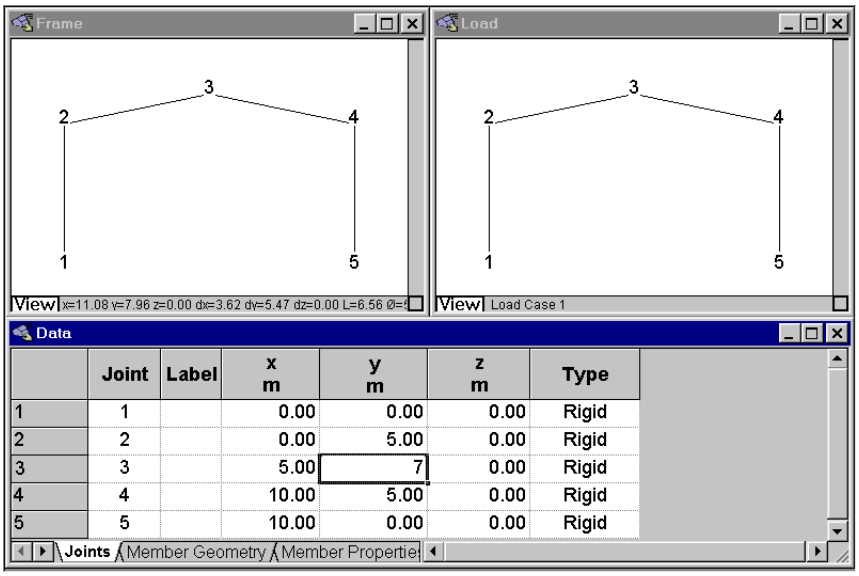
To change a coordinate of a joint

- **Make sure the Data window is in front**
- **Click on the coordinate to be changed**



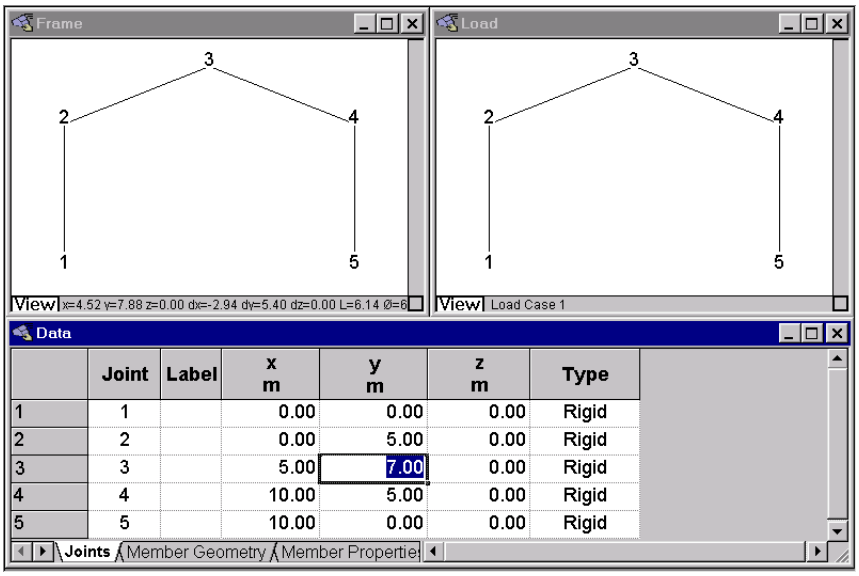
	Joint	Label	x m	y m	z m	Type
1	1		0.00	0.00	0.00	Rigid
2	2		0.00	5.00	0.00	Rigid
3	3		5.00	6.06	0.00	Rigid
4	4		10.00	5.00	0.00	Rigid
5	5		10.00	0.00	0.00	Rigid

- **Type in the new value for the coordinate**



	Joint	Label	x m	y m	z m	Type
1	1		0.00	0.00	0.00	Rigid
2	2		0.00	5.00	0.00	Rigid
3	3		5.00	7	0.00	Rigid
4	4		10.00	5.00	0.00	Rigid
5	5		10.00	0.00	0.00	Rigid

- **Press the Enter key or click inside the Frame or Load window on Windows**



	Joint	Label	x m	y m	z m	Type
1	1		0.00	0.00	0.00	Rigid
2	2		0.00	5.00	0.00	Rigid
3	3		5.00	7.00	0.00	Rigid
4	4		10.00	5.00	0.00	Rigid
5	5		10.00	0.00	0.00	Rigid

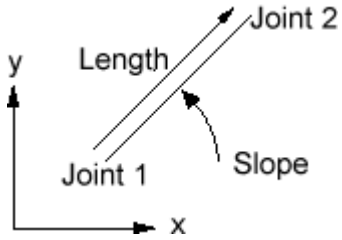
The structure will be re-drawn in the Frame and Load windows to reflect the changes you have made. You can repeat this process for any of the other joints in the structure. You can use the Tab and Return keys or the arrow keys to move from number to number in the table or you may simply click on the number you wish to change.

Note that changing the coordinates of a joint so that it is the same as another joint in the structure will not create a connection between the members at that joint. A connection can only be made by dragging one joint onto the other in the Frame window.

If you double click on a joint in the Frame window, Multiframe will display a dialog that allows you to change the coordinates of the joint.

Changing the position of a joint will change the length and/or slope of any members that are connected to it. The length and slope of the members in the structure may also be changed directly. The Member lengths and slopes may be displayed by choosing the Member Geometry command in the Data sub-menu under the Display menu.

Simply click on the numbers to be changed and type in the new values. The drawing in the Frame and Load windows will be updated to reflect your changes. Remember that the slope of a member is measured in degrees with positive angles being measured from a zero angle on the horizontal plane passing through the lower joint. Joint 1 will be the leftmost joint in the case of a horizontal or sloping member as viewed in the front or right side views or the bottom-most joint in the case of a vertical member.



You can also change the length and slope of a member by double clicking on it in the Frame window and typing in the new values in the dialog that appears.

If you change the length of a member, Joint 1 will be held fixed and Joint 2 will be moved to give the member the required length. The slope will not be changed. Similarly, changing the slope of a member will move Joint 2 and will leave Joint 1 in the same position while leaving the length of the member unchanged.

Joint and Member Numbers

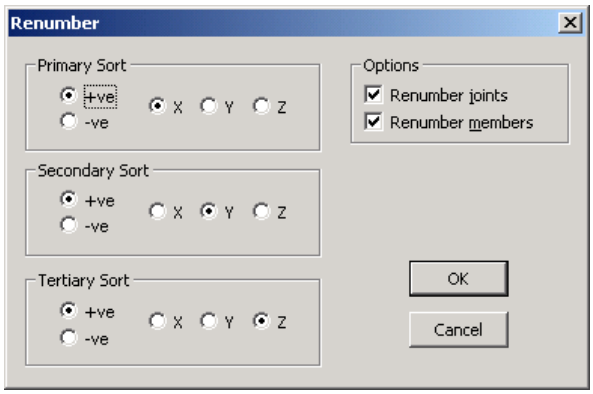
As you add members and joints to a structure, Multiframe will automatically assign numbers to them. If you wish to renumber the joints and or members, you can do so using the Renumber command from the Geometry menu.

Renumbering is performed by sorting the joints by their coordinates. Three levels of sorting are used to perform the renumbering. The primary sort is used to first sort joints along a specified direction. All joints with the same location in the primary sort direction are then sorted using a secondary sort. Finally, all joints having the same location in both the primary and secondary sort directions are then sorted using the tertiary sort.

In 3D, the primary sort can be visualised of as sorting all the joints into planes of joints, the planes being aligned perpendicular to the sort direction. The secondary sort then takes the joints in each of these planes and sorts them into lines of joints, the lines being aligned perpendicular to the secondary sort direction. The tertiary sort then performs a simple sort to order the joints along each the lines of joints in each of the planes.

To renumber the structure

- **Choose Renumber from the Geometry menu**



- **Choose the Primary, Secondary and Tertiary sort directions**
- **Click on the OK button**

Note that you can renumber after analysis and your results will be preserved but displayed using the new joint and member numbers.

You can display the joint and member numbers on the members in the drawing windows by using the Symbol command from the Display menu and turning on Joint Numbers and/or Member Numbers.

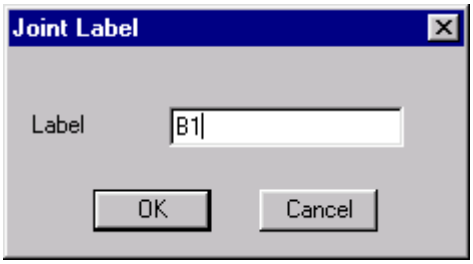
Joint and Member Labels

Joints and members within in structure may be labelled with a text string of up to 15 characters. Labels can be used for many purposes such as identifying the position of a member by floor or grid location, identifying section types with a legend that would be used on a marking plan, or to provide an alternative numbering scheme that remains unchanged as the structure is modified.

To set the label on a joint or a member

- **Select the joint(s) or member(s)to be labelled**
- **Choose Joint Labels or Member Labels from the Frame menu**

A dialog box will appear with a field in which to type the label



- **Type in the text for the label.**
- **Click on the OK button**

You can display the joint and member labels on the members in the drawing windows by using the Symbol command from the Display menu and turning on Joint Labels and/or Member Labels.

Restraints

A structure must be sufficiently restrained for analysis to be carried out. A minimum requirement is that the structure be restrained against movement in each of the x, y and z directions and that it should not have any mechanisms caused by collapsible pinned structures.

The restraints on the structure are specified by first selecting the joint or joints to be restrained and then selecting the appropriate type of restraint. The restraint may take the form of a zero displacement restraint or a spring. A joint may have either a restraint, or a spring for each degree of freedom but not combinations of these restraints for any degree of freedom. Thus a joint could have a vertical restraint and a horizontal spring but not a horizontal restraint and a horizontal spring or a vertical restraint and a vertical spring.

All restraints and springs that are applied to a joint are specified in the local coordinate system at each joint. The local coordinate system is typically the same as the global coordinate system but can be modified using the Joint Orientation feature described later in this chapter.

See the Prescribed Displacement description in the Loads section for information on applying non-zero displacements.

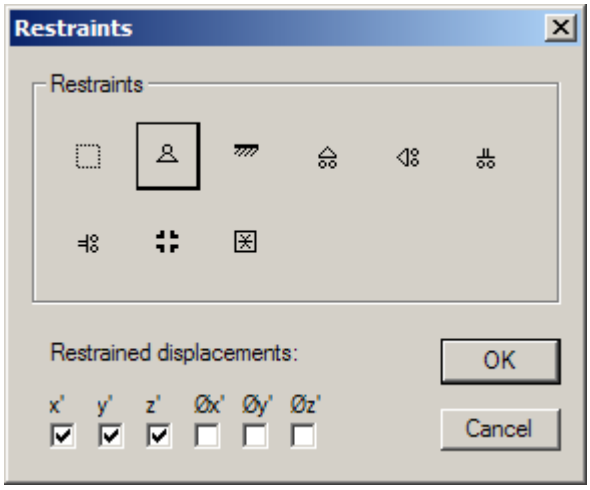
You can modify the restraint or spring on a joint by double clicking on the restraint or spring icon at the joint (and not on the joint itself).

Joint Restraint

To specify a restraint at a joint

- **Select the joint or joints to be restrained**
- **Choose Joint Restraint from the Frame menu**

A dialog box will appear with a range of icons for the various types of restraint



- **Click on the appropriate restraint icon**
- **Click on the OK button**

Selecting the "no restraint" icon (the first icon in the list) will remove all joint restraints from the selected joints. Selecting any other icon will remove any prescribed displacements or springs from the selected joints and replace them with the selected restraint.

Each of the possible restraints is indicated by an icon. As you click on the icons, the check boxes at the bottom of the dialog will display the degrees of freedom that will be restrained if you select that icon.

You can also click on the check boxes directly to choose which degrees of freedom you wish to restrain. If the combination of restraints you select is not one of the standard icons, Multiframe will display the special restraint icon (the last icon in the list).

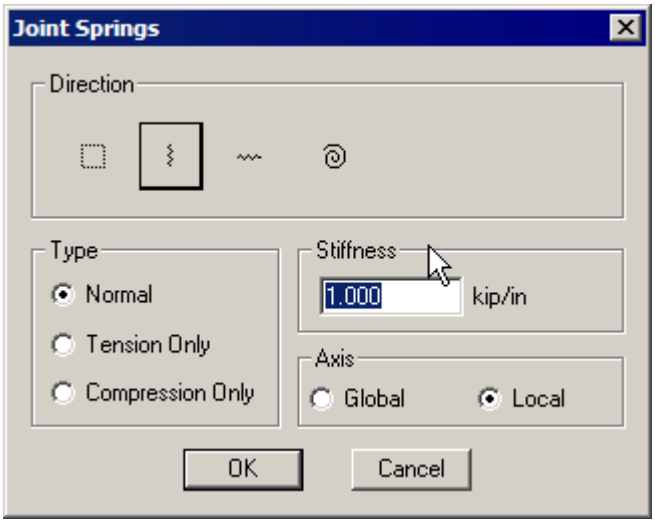
After you have added a restraint to the frame, you can change its properties by double clicking on the restraint icon in the Frame window. You can also edit the individual degrees of freedom of the restraint in the Restraints table in the Data window.

Springs

To attach a spring support to a joint

- **Select the joint or joints supported by springs**
- **Choose Joint Spring from the Frame menu**

A dialog box will appear with icons for the spring directions.



- **Click on the icon which shows the direction in which the spring is to act**
- **Type in a value for the spring constant**
- **Choose the axis system of the spring.**
- **Select the type of spring.**
- **Click on the OK button**

Selecting the "No spring" icon (the first icon in the list) will remove all springs from the selected joints. Selecting any other icon will remove any restraints or prescribed displacements with the same degree of freedom (if any) from the selected joints and replace them with springs. The rotational spring icons each have an axis drawn through them in the 3D view that indicates the axis about which they provide rotational stiffness.

Springs acting in a local axis system are aligned to the local axis system of the node to which they are attached.

Springs may also be defined as tension or compression only. As the behaviour of the spring is no longer linear, only a nonlinear analysis considering tension/compression only effects will correctly model these types of springs. In all other analyses the springs will be treated as a normal spring with no tension/compression only effects.

All restraints, springs and prescribed displacements that are applied to a joint are specified in the local coordinate system at each joint. The local coordinate system is typically the same as the global coordinate system but can be modified using the Joint Orientation feature described later in this chapter.

After you have added a spring to the frame, you can change its properties by double clicking on the spring icon in the Frame window. You can also edit the individual stiffness's of the spring in the Springs table in the Data window.

Editing Restraints Numerically

You can display and edit tables of springs and restraints in the Data window. You can use the commands from the Data sub-menu under the Display menu to control which table is on display at any time.

To change the value of a restraint or spring

- **Click on the number you wish to change**
- **Type in the new value**
- **Press the enter key**

While editing numbers in the Data window you can use the scroll bars to scroll through the list of data, horizontally or vertically, to find the number you want to alter. If you wish, you can use the Return or Enter, Tab or arrow keys to move to another number rather than pressing enter.

You can resize the columns in the table by dragging the lines that separate the column titles. You can change the text font and size used in the table by using the Font… command from the View Menu while the Data window is in front on the screen. The format of the numbers displayed in the table can be controlled by using the Numbers… command from the View menu while the Data window has focus.

You can also double click on the restraint icon at a joint (but not the joint itself) to modify that restraint.

Grouping

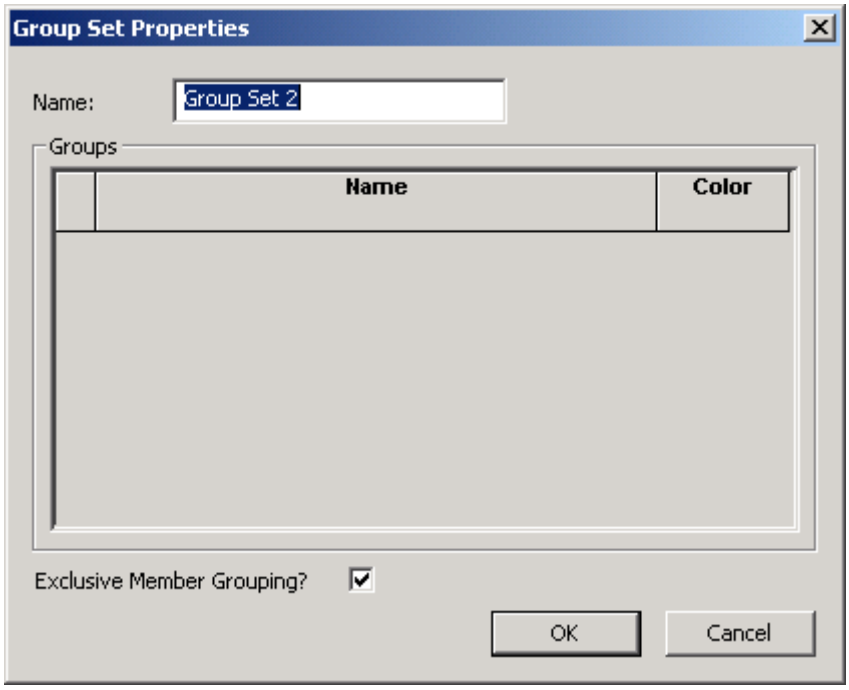
Multiframe allows you to arbitrarily group members together so that they can readily be selected and identified within the frame. The implementation of grouping within Multiframe is hierarchical and organises groups into sets of groups or Group Sets. This allows groups to be organised into sets based upon a common categorisation. For example, groups used to define floors within a building can be organised into a Floors group set.

Group Sets

Group sets contain a list of groups and provide a means of organising groups into logical sets based upon a common criterion. The grouping of members within a group can be set to only allow members to be associated with a single group within the group set so that members are exclusive to a single group within the set. By default, all frames contain a single group set.

Only a single group set is active within the user interface at any time. The user can select which group set is active from the Current Group Set submenu contained within the Group menu. Alternatively it can be selected using the Group toolbar. When a new group set is added to the model it is set as the current group set. To add a group set

- **Choose Add Group Set… from the Group menu**



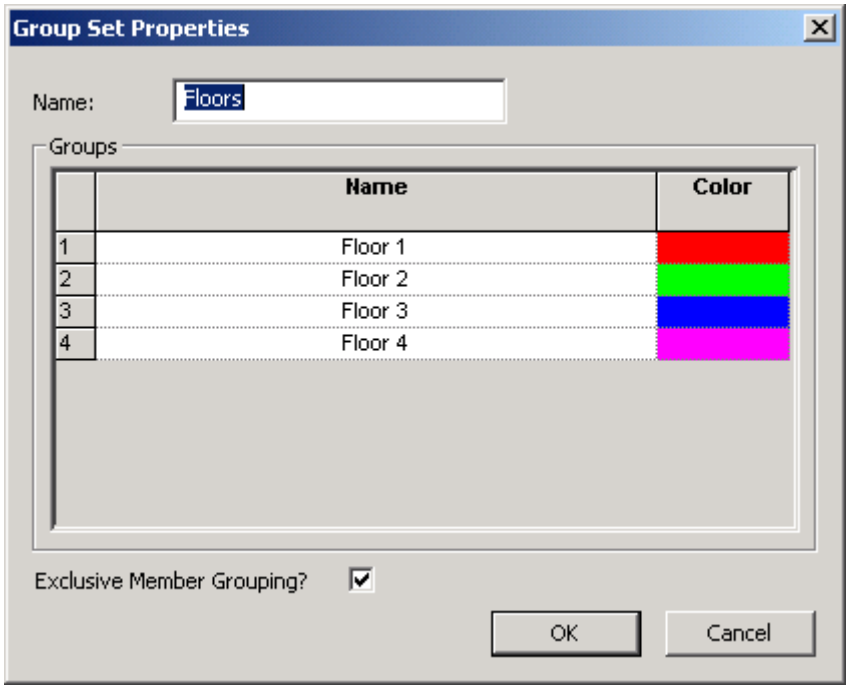
- **Type in the name of the group set.**
- **Select the Exclusive Member Grouping checkbox if members are only allowed to be contained within a single group within the group set .**
- **Click on the OK button**

Groups within the current group set can be displayed within the Frame Window by choosing to draw members using group colours. All grouped elements will be displayed in the colour of the group in which they are contained. Any ungrouped members will be displayed in black. The choice to draw members using group colours is specified via the Symbols dialog.

A group set may be edited to change its names, the member exclusivity and the properties of the groups contained within the set. To edit a group set

- **Choose Edit Group Set… from the Group menu**

The Group Set properties dialog will be displayed that lists the groups contained within the current group set and the properties of the groups contained within the set.

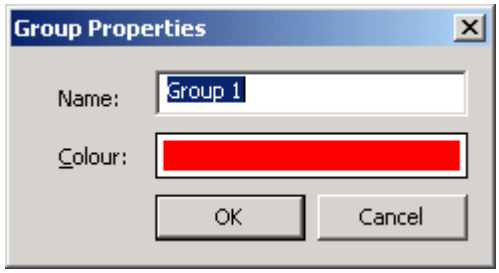


Groups

A group defines an arbitrary list of members and joints. It provides a simple way of selecting and interacting with the grouped members or nodes. Each group has a name and colour that identify the group, both of which are set by the user. When a group is contained within a group set using exclusive member grouping, the members with the group cannot be part of another group with the same set.

To create a group of members

- **Select the members to be groups in any of the graphical views or the tables listing members.**
- **Choose Add Group… from the Group menu**



- **Type in the name of the group.**
- **Select the colour to be associated with the group. Double clicking on the colour field will open the colour picker dialog.**
- **Click on the OK button**

If the members are displayed using group colours, the selected members will be displayed using the colour associated with the group. The properties of a group may be edited via the table in the Group Set dialog.

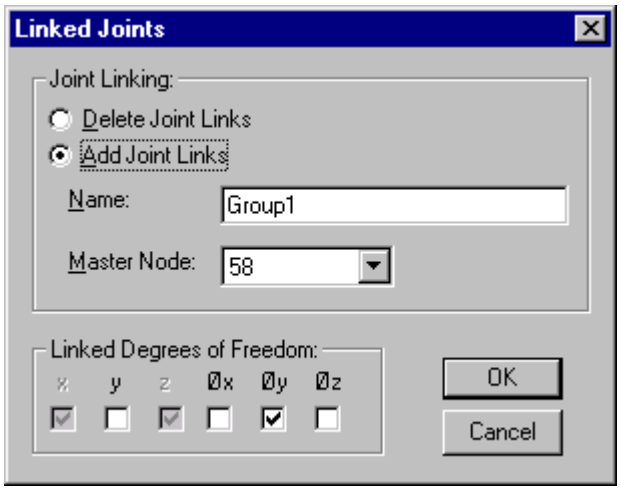
Linking Joints Or Master-Slave

Multiframe allows you to rigidly link groups of joints together so that they move together in response to either static or dynamic loads. This has a twofold benefit. Firstly, it allows you to simulate rigid structural elements such as a floor slab, and secondly it significantly reduces the size of the stiffness matrix resulting in lower memory requirements and faster analysis times.

Joints can be linked together on a degree-of-freedom basis. This means some degrees-of-freedom can be linked together and others left independent. In the case of a rigid floor slab for example, all the joints on that level could have their x, z and Øy deflections linked together to simulate the rigid translation and rotation of the slab while still allowing bending of the floor. When linking a rotational degree of freedom, Multiframe automatically selects the corresponding displacement degrees of freedom that must also be imposed to enforce rigid body motion in the plane of the rotation.

To rigidly link a group of joints

- **Select the joints to be linked in the Frame window**
- **Choose Joints Linking from the Frame menu**

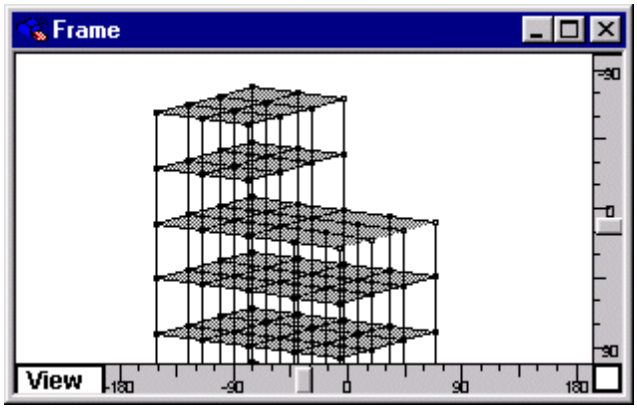


- **Select a master joint.**

In most cases the default master joint selected by Multiframe will be adequate. It is only when linked group intersect that the Master Joint would have to be changed to a joint contained in both linked groups of joints.

- **Type in a name for the group and select the degrees of freedom to be linked**

The linked group of joints is displayed as a shaded area in the Frame window. The Master Joint in each group of linked joints is also highlighted using circle of darker shading.



The properties of a linked group of joints can be edited by double clicking on the linked region displayed in the Frame window.

The linked joints are also displayed in tabular form in the Data window. Choose Linked Joints from the Data sub-menu to display this table or click on the Linked Joints tab at the bottom of the Data Window. The rows of this table corresponding to the Master Joints are shown in bold.

Data

	Group	Joint	Label	x	y	z	Øx	Øy
1	**Group 1**	**6**		**Linked**	-	**Linked**	-	**Linked**
2	Group 1	7		Linked	-	Linked	-	Linked
3	Group 1	8		Linked	-	Linked	-	Linked
4	Group 1	9		Linked	-	Linked	-	Linked
5	Group 1	10		Linked	-	Linked	-	Linked
6	Group 1	31		Linked	-	Linked	-	Linked
7	Group 1	32		Linked	-	Linked	-	Linked
8	Group 1	33		Linked	-	Linked	-	Linked
9	Group 1	34		Linked	-	Linked	-	Linked
10	Group 1	35		Linked	-	Linked	-	Linked
11	Group 1	62		Linked	-	Linked	-	Linked
12	Group 1	63		Linked	-	Linked	-	Linked

Linked Joints | Springs | Joint Masses

The linking is defined for the global XYZ directions. If you link two or more joints together in the X direction then they will be locked together for any movement in the global X direction.

The following restrictions apply when linking joints and to the restraints, or supports, attached or applied to the linked joints.

- **Joints may be common to several groups of linked joints provided that the master joint in all of the intersecting groups is the same joint.**
- **Prescribed displacements can only be applied to linked joints that are master joints.**
- **Restraints to linked degrees of freedom must be applied to the master joint**
- **The reactions in the direction of the linked degrees of freedom for a group of linked joint are computed at the master joint.**

The compliance of the frame to these rules is tested when the user analyses the frame.

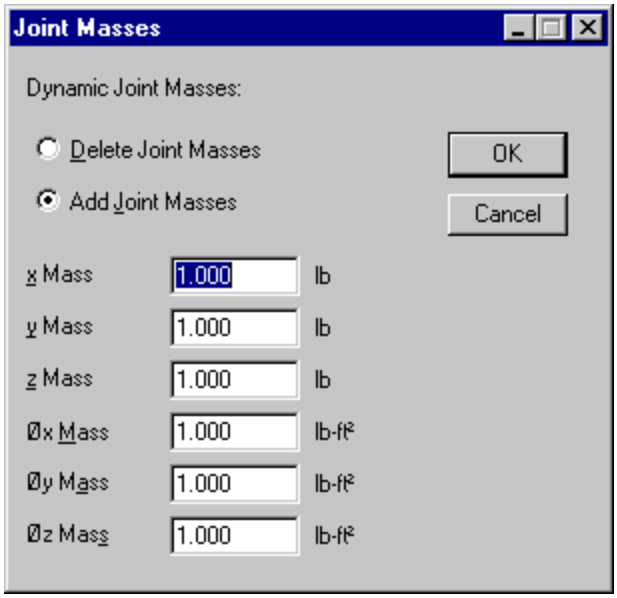
The joints linked together in a linked group may be selected using the Linked Joints command in the Select menu. From version 8.6 of Multiframe the elements selected when the joints are linked are also stored with the linked group. These will also be selected when the Select Linked Joints command is executed.

Joint Mass

If you are performing a dynamic analysis using Multiframe4D, you may wish to add additional masses to the structure to simulate the effects of equipment or construction loading which will affect the inertia of the frame. The Joint Mass command from the Frame menu allows you to add these additional masses at joints in the structure.

To add a mass at a joint or joints

- **Select the joint or joints**
- **Choose Joint Mass from the Frame menu**



- **Enter the mass values to be associated with each global direction of movement**

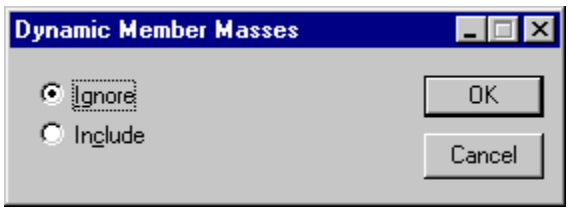
If you have already added masses that you wish to remove, you can click the Delete Joint Masses radio button to remove Joint Masses from the selected joints.

Member Masses

If you are performing a dynamic analysis using Multiframe4D, you may wish to include or ignore the masses of the members in the frame and the effect they will have on the inertia of the frame. The Member Mass command from the Frame menu allows you to ignore or include the effect of member mass in dynamic analysis.

To specify whether member mass is to be included

- **Select the member or members**
- **Choose Member Masses from the Frame menu**



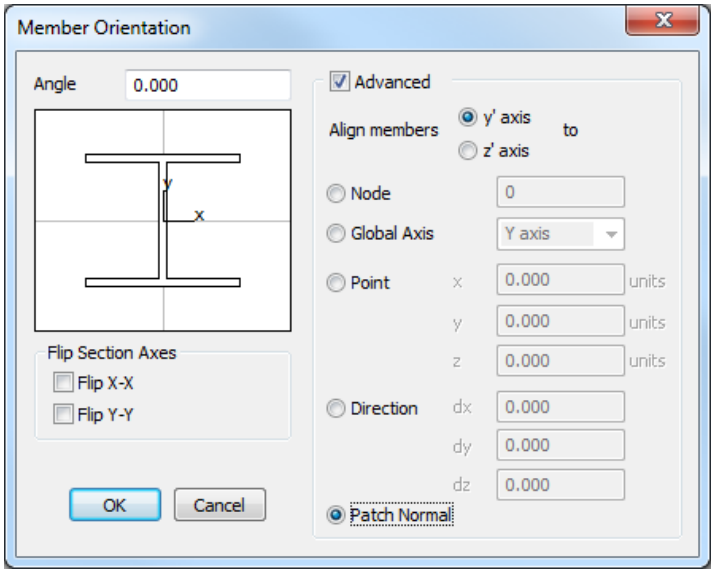
- **Click Ignore or Include as appropriate**

Section Orientation

When analysing a three dimensional frame it is necessary to know the orientation of the structural section used for each member relative to the global coordinate system. Initially, Multiframe assumes that the web or direction of principal strength of the section is aligned so that the principal direction lies in a vertical plane passing through the member. In the case of a vertical column, it is assumed that the principal direction passes through the global x axis. If you wish to have the section oriented at another angle (referred to by some engineers as the beta angle) you can use the Member Orientation… command from the Frame menu.

To change the orientation of a member or members

- **Select the member or members to be specified**
- **Choose Member Orientation from the Frame menu**



A dialog box will appear which displays the section's shape. The view of the section shape in the dialog is the view you would see if your eye was at joint 2 and you were looking down the member towards joint 1.

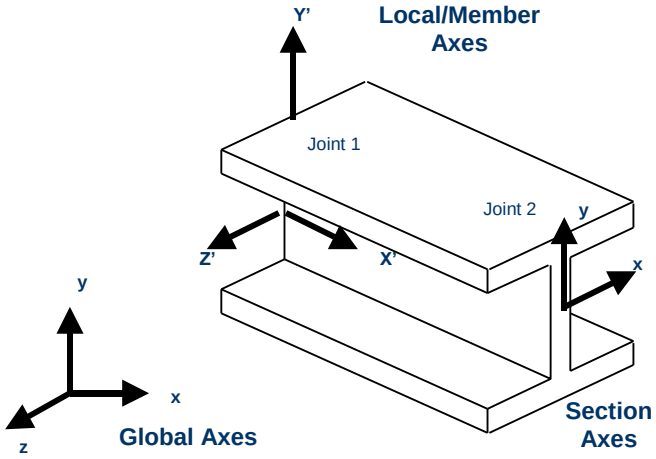
- **Click and drag on the shape to rotate it to a new orientation or type in a new orientation angle**
- **Click OK to set the new orientation**

When you set the orientation you are effectively defining the direction of the local y' axis of the member. The angle you type is the angle between the y' axis and a vertical plane passing through both ends of the member. As the angle increases the y'-axis will rotate towards the z'-axis.

A number of parametric options are also available for specifying the orientation of members. Instead of specifying an angle, the user can choose a number of advanced options that define the plane containing either the member's y-axis or z-axis. The plane is defined by the local x-axis of a member and either a point in space or a direction. The orientation point can be defined by its coordinates or by specifying a joint. Similarly, the direction may be defined as either a vector or by specifying a global axis direction. Care must be taken to ensure that the point or direction define a valid plane. As such the orientation point should not lie along the line of the x-axis and the orientation direction should not be parallel to the members x-axis.

If a member is attached to a patch it is possible to automatically align the member orientation so that it is perpendicular to the patch, on the opposite side to the patch normal. If you wish to align the member to the patch normal, then select Patch Normal and then click Flip X-X and the section will be rotated by 180 degrees. If one or more of the selected members are not attached to a patch then a warning message will be shown. If a member is attached to more than one patch ie it is on the edge where two patches join, it will be aligned to the first patch detected.

You can also choose to flip the orientation of the section about its x and/or y axes.



You can display the section axes on the members in the Frame window by using the Symbol command from the Display menu. You can also show the member axes in this way.

Automatically Align and Offset Member to Patch

The Member Align and Offset to Patch command will automatically align a member to be perpendicular to the patch it is attached to. It is offset so that the bottom of the section is flush with the patch.

- **Select one or more members that are attached to a patch**
- **Choose Member Align and Offset to Patch… from the Frame menu**

The member will be orientated perpendicular to the patch on the opposite side to the patch normal. If you wish to align the member to the patch normal, then do the above and then open the Member Orientation Dialog (Frame > Member Orientation) and click Flip X-X and the section will be rotated by 180 degrees.

If one or more of the selected members are not attached to a patch then a warning message will be shown.

If a member is attached to more than one patch ie it is on the edge where two patches join, it will be aligned to the first patch detected.

Please note that the member will be offset to its base plus half the bending thickness of the patch. In the render mode the patch thickness is not drawn so the member will appear a small distance from the patch.

Section Properties

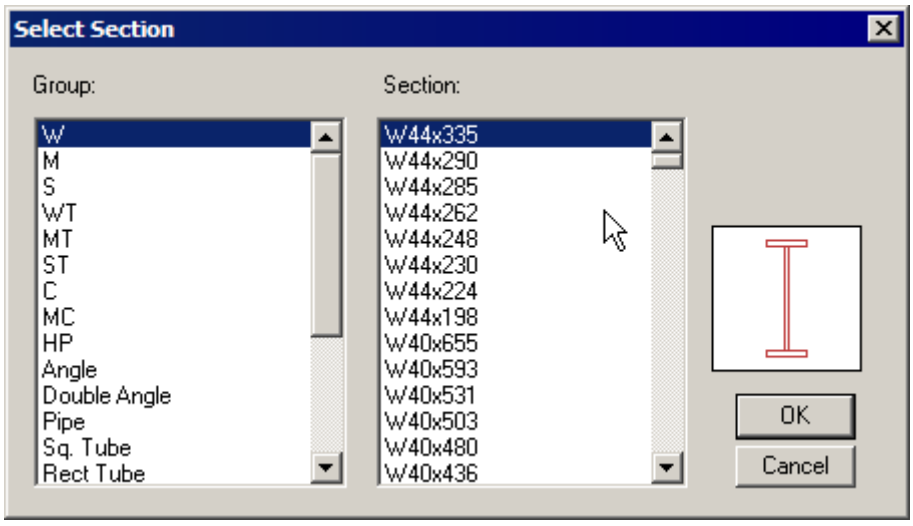
To compute the deflections in the structure it is necessary to know the geometric and material properties of the members that make up the structure. Multiframe has a library of pre-defined section properties for the most commonly used steel sections. If you wish to use structural sections other than those pre-defined in the Sections Library, see "Adding a custom section" below.

Note that Multiframe uses the word "Member" to describe a geometric component of a structure such as a beam, column or strut and the word "Section" to describe a particular structural shape or section with its own material and geometric properties. Each member of the structure is constructed using a particular section.

To specify the section for a member

- **Select the member or members to be specified**
- **Choose Section Type from the Frame menu**

A dialog box will appear with a list of the groups and sections available in the Sections Library.



- **Click on the name of the group you wish to choose from**
- **Click on the name of the section you wish to choose**
- **Click the OK button**

You must specify the section type for every member in the structure before carrying out the analysis.

You can display the sectional shapes on the members in the Frame window by using the Symbol command from the Display menu and turning on Section Shapes.

Adding a Standard Section

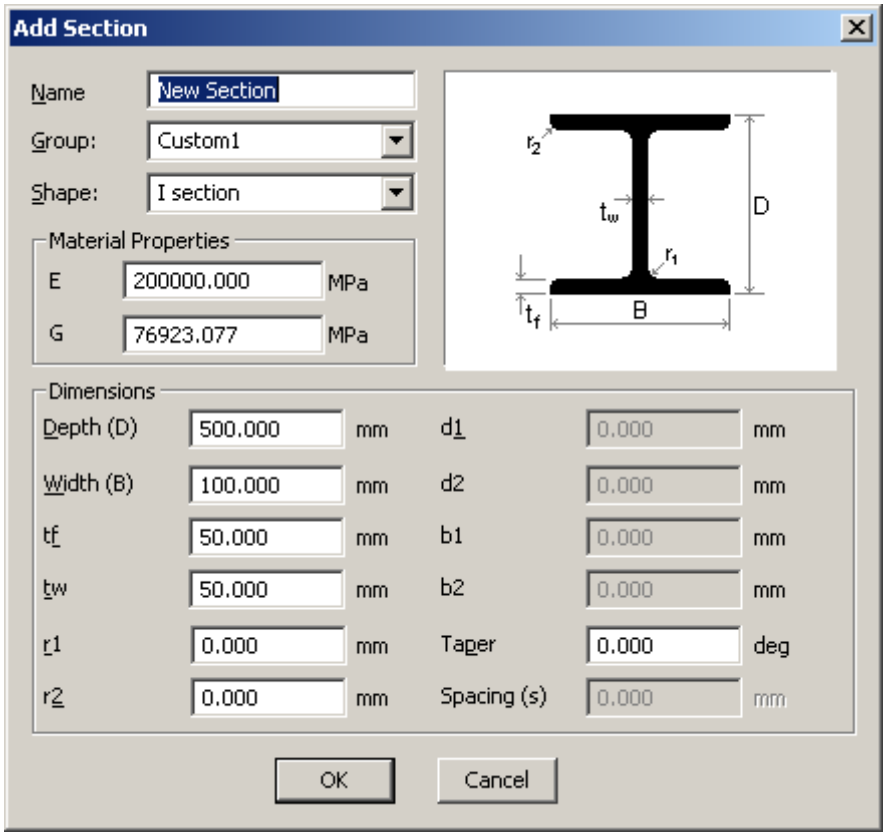
If the structural section you require is not contained in the Sections Library, you can define your own section and store it in the library or store it with the structure. If the shape of the section is one of the standard shapes supported within Multiframe then the section can be added by specifying the dimensions of the shape and Multiframe will compute the properties of the section. Adding sections with non-standard shapes is described in the next section.

The sections are arranged in the library in groups. Each of the groups usually consists of a range of sections of a similar type. This corresponds to the various tables of sections found in engineering handbooks. At the end of the list of groups are a number of groups labelled Custom1, Custom2, Custom3 and Frame. These groups are designed to be used to store special sections in.

To add a standard section

- **Choose Add Standard Section from the Sections submenu under the Edit menu**

A dialog box will appear with a list of group names, a field for the section's name, a list of icons to indicate the sections shape and a number of fields in which to input the material properties and dimensions of the section.



- **Type in a name for the section**
- **Choose the name of the group you wish to store the member in from the Group pop-up menu**
- **Choose the shape of the custom section from the shape pop-up menu**
- **Fill in the fields in the Material Properties group with the value of Young's modulus and the Shear Modulus.**
- **Fill in the active fields in the Dimensions group to specify the size of the shape.**

If your section does not have one of the shapes shown in the list, leave the Unknown section options selected.

- **Click on the OK button**

Multiframe will now add the section to the library, the properties of the section will be automatically be computed.

Only the groups in the library that are not locked will be shown in the list of groups. If you wish to store the section in the library and have it available for use in other structures, store the section in one of the groups other than the group named Frame. Sections stored in the Frame group will be stored with the structure and will not appear in the list unless you are using this structure. You will probably find it convenient to store most of your sections in the Frame group to avoid cluttering up your library with sections that are only used in one or two structures.

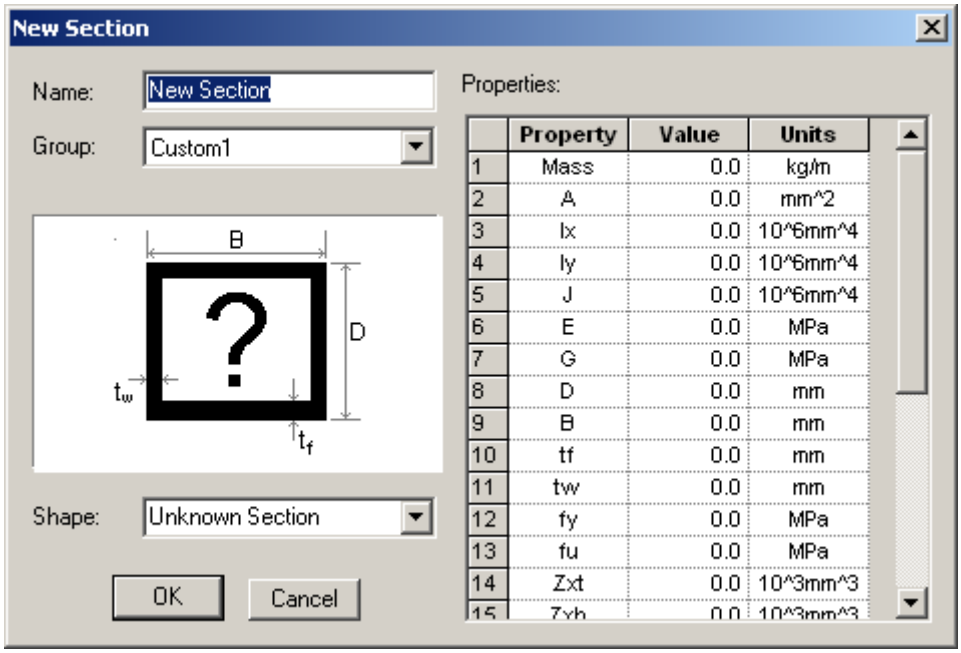
Adding a Custom Section

If the structural section you require is not contained in the Sections Library, and is not one of the standard shapes supported by Multiframe, you can define the section by specifying all the key cross section properties. Although you may find it easier to use the Section Maker application to create and install your section, this section describes how you can add a section by simply typing in the key sectional properties.

To add a custom section

- **Choose Add Section from the Sections submenu under the Edit menu**

A dialog box will appear with a list of group names, a table of section properties, a field for the section's name and a list of icons to indicate the sections shape.



- **Type in a name for the section**
- **Choose the name of the group you wish to store the member in from the Group pop-up menu**
- **Fill in the fields in the table with appropriate values using the down arrow key to move from field to field**
- **Choose the shape of the custom section from the shape pop-up menu**

If your section does not have one of the shapes shown in the list, leave the Unknown section options selected.

- **Click on the OK button**

Note that it is not necessary to fill in all the fields if you do not wish to use the variables later in the CalcSheet. However, the following fields must be filled in:

Mass/Weight	Mass per unit length (if using Self Weight)
A	Cross sectional area
Ix	moment of inertia about the major(x-x) axis
Iy	moment of inertia about the minor (y-y) axis
J	Torsion constant
E	Young's Modulus
G	Shear Modulus

If you which to view stresses on your members, you should also fill in the following fields

Sxt (Zxt)	Elastic section modulus about x at top side of section
Sxb (Zxb)	Elastic section modulus about x at bottom side of section
Syl (Zyl)	Elastic section modulus about y at left side of section
Syr (Zyr)	Elastic section modulus about y at right side of section

If you wish to view rendered shapes of your section, you should also fill in the following fields

D	Depth of section
B	Breadth (width) of section
tf	Flange thickness
tw	Web thickness

If you wish to carry out analysis taking into account deflection due to shear effects, you should also fill in the following fields

Asx	Shear area in x direction
Asy	Shear area in y direction

Be careful to ensure that the units of the values you enter match those of the fields shown in the table on screen.

Only the groups in the library that are not locked, will be shown in the list of groups. If you wish to store the section in the library and have it available for use in other structures, store the section in one of the groups other than the group named Frame. Sections stored in the Frame group will be stored with the structure and will not appear in the list unless you are using this structure. You will probably find it convenient to store most of your sections in the Frame group to avoid cluttering up your library with sections that are only used in one or two structures.

Note

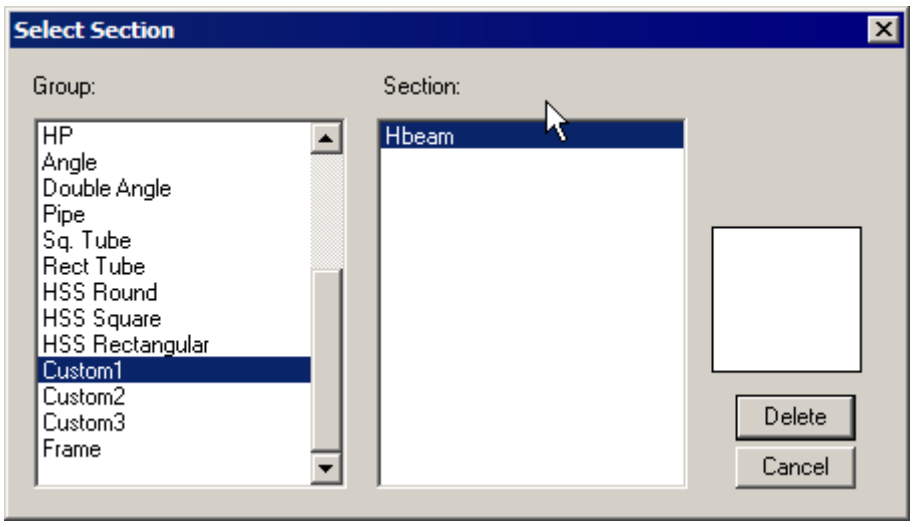
Each time Multiframe starts up, it will look for the default section library c:\program files\multiframe\sectionslibrary.slb file. Any changes to the sections library will automatically be saved to this file.

Removing a Section

To remove a custom section from the library

- **Choose Delete Section from the Sections menu under the Edit menu**

A dialog box will appear with a list of groups and sections.



- **Click on the group and section to be deleted**
- **Click on the Delete button**

Editing a Section

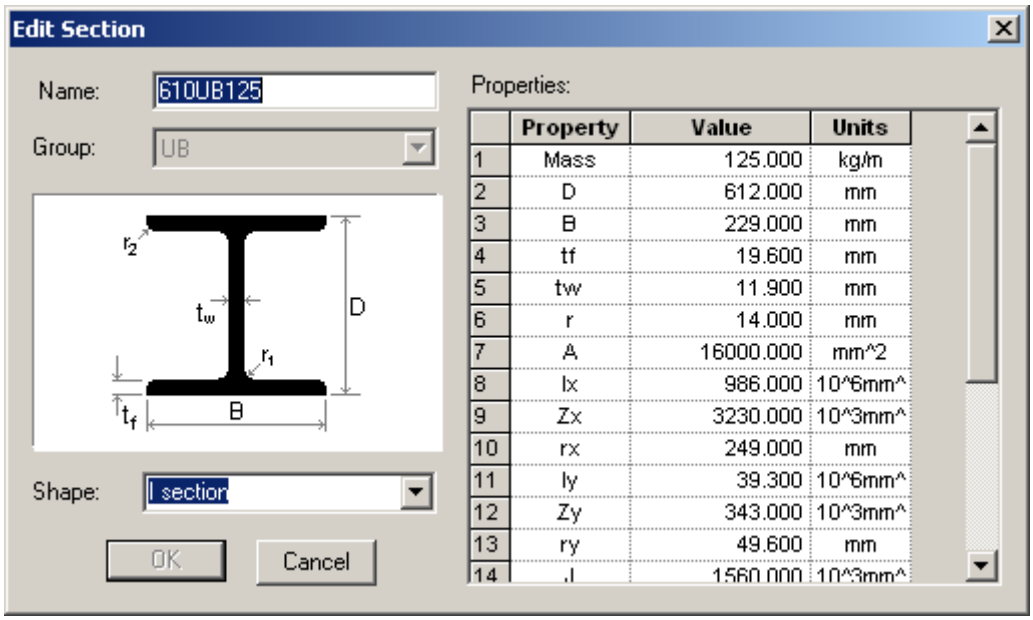
To change the section properties of a custom section

- **Choose Edit Section from the Sections menu under the Edit menu**

A dialog box will appear with a list of groups and sections.

- **Click on the group and section to be edited**
- **Click on the Edit button**

A dialog box will appear with the data for the section shown in a table.



- **Enter the information you wish to change using the Tab key to move from field to field**
- **Click on the OK button**

You can also use this procedure to look at the actual values stored in the library for any section without changing them. Simply click the Cancel button after viewing the properties of interest to you.

If the OK button is drawn in grey this indicates that the group this section is stored in is locked and the section properties cannot be changed.

Section Type

You can review the sections used in a structure by displaying the Sections Table in the Data window. Choose Sections from the Data sub-menu under the Display menu to display this table. The table of sections displays details all of the sections used in the frame. It contains information about the sectional properties as well as the dimensional information about the sections. This is particularly useful when copying information to a spreadsheet for further processing (see Appendix E for more information).

Materials

In Multiframe the materials properties used in the analysis and design of a member can be defined in number of ways. In older versions of Multiframe, materials properties for design (i.e. E and G) were stored as section properties and as such it was only necessary to allocate a section type to a member in order to define it's materials properties.
In the current version of Multiframe a material may now be assigned to members by the user. However, this is often unnecessary as a material will be automatically assigned to a member when a section type is specified as sections in the library now store a material which is used as the default material for members in Multiframe. Note that this default material associated with the section will only be assigned to the member if a material has not been previously assigned to the member.

Note (Older Libraries and Models)

When using older libraries and frames, the elastic material properties required for analysis will still be obtained from the section properties.

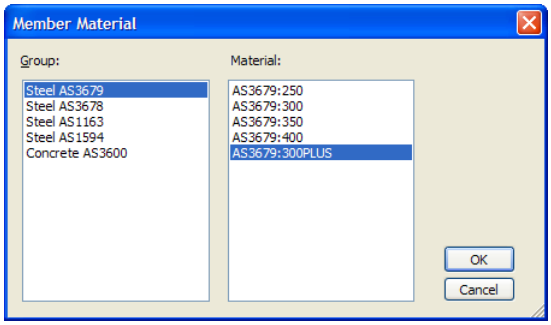
Member Materials

The Sections Libraries provided with Multiframe also have a list of pre-defined properties for the most commonly used materials. If you wish to use materials other than those pre-defined in the Sections Library, see "Adding a Material" below.

To specify the material for a member

- **Select the member or members to be specified**
- **Choose Member Material from the Frame menu**

A dialog box will appear with a list of the groups and materials available in the Sections Library.



- **Click on the name of the group you wish to choose from**
- **Click on the name of the material you wish to choose**
- **Click the OK button**

A material does not need to be assigned to members. With older versions of the sections library the material properties required for analysis are stored with the sections. In this situation, if no material is assigned to a member then the material properties stored with the sections will be used. For updated libraries that contain groups of materials, a material will be associated with each of the sections and this will be used as the default material for a member and is assigned to the member when the section type is selected. Note that if a material has already been allocated to a member then the material is not re-assigned when the section type is modified.

The materials associated with the members can be displayed in the Frame window by using Member Legend to display members according to material.

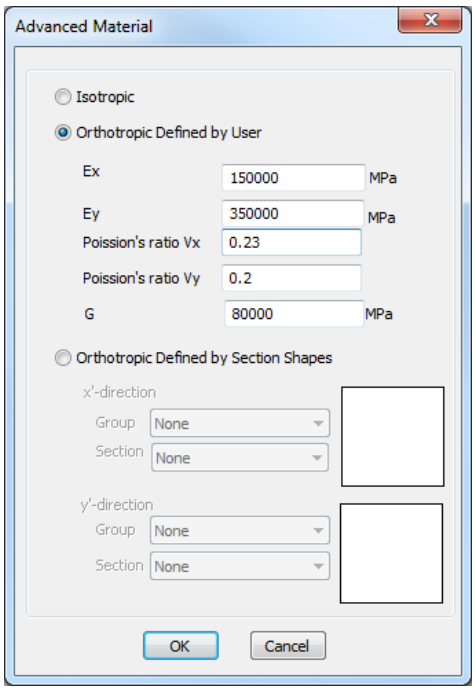
Patch Orthotropic Materials

Multiframe extends its support to orthotropic materials for plates. This will be done through “Advanced material button” in patch property dialog window.

To apply orthotropic material to a patch

- **Select the patch to be edited**
- **Choose Patch Properties menu or short cut menu**

- **In material group, click "Advanced" button and a dialog box will appear with advanced material property**



- **Select material type & Type in values for the material**
- **Click on the OK button**

Note that when working in dialog boxes, you can use the Tab key to move from one editing box to the next.

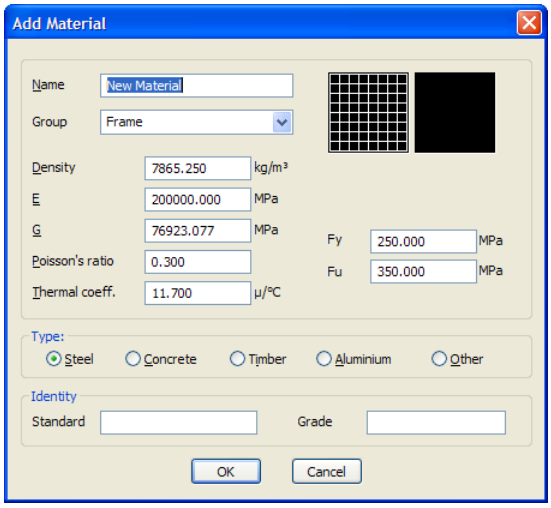
Adding a Material

If the material you require is not contained in the Sections Library you can define the material by specifying all the material properties.

To add a material

- **Choose Add Material from the Materials submenu under the Edit menu**

A dialog box will appear with a list of group names, fields for the material's name and the properties of the material. The dialog also provides fields to specify the type, colour and a reference to standard that defines the material.



- **Type in a name for the material**
- **Choose the name of the group you wish to store the material in from the Group pop-up menu**
- **Fill in the fields specifying the properties of the material.**
- **Choose the type of the material.**

Be careful to ensure that the units of the values you enter match those of the fields shown in the table on screen.
Only the groups in the library that are not locked, will be shown in the list of groups. If you wish to store the material in the library and have it available for use in other structures, store the section in one of the groups other than the group named Frame. Materials stored in the Frame group will be stored with the structure and will not appear in the list unless you are using this structure. You will probably find it convenient to store most of your materials in the Frame group to avoid cluttering up your library with materials that are only used in one or two structures.

Editing and Deleting Materials

The properties of materials can be modified and materials may be removed from the library using the commands in the Materials submenu under the Edit Menu.

Working with the Sections Library

When working with lots of different sections for different projects, some thought has to be given to how to manage the section libraries. The best way to work with the section library is dependent on the number of different projects (simultaneously) and whether the projects are done in a multi-user network environment or not.

Approach 1. - Default

Sections Library is in same directory as Multiframe program
Use Section Maker to add custom sections and groups
Add add-hoc sections to custom groups or to Frame group

Approach 2. - Project based

Rename default library
Copy default library to project folder at start of each project
Optional - rename sections library to project name

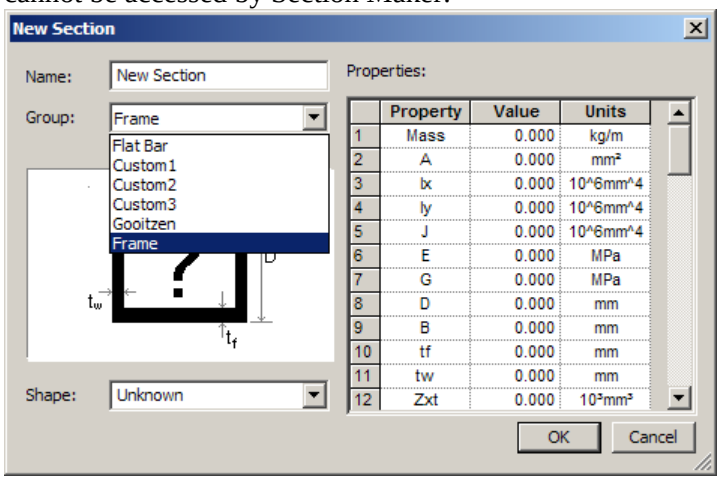
Load appropriate library each time Multiframe starts up
Back up sections library with project data files

Tip 1. – Renaming the default library

By changing the name of the default section library, Multiframe will prompt you which library you wish to use each time it starts up. The default section library is: c:\program files\multiframe\sectionslibrary.slb.

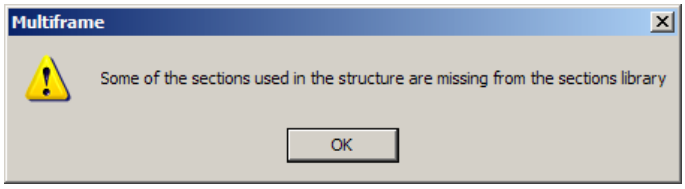
Tip 2. – Frames Group

For sections that will only be used in one particular project and to prevent cluttering the library too much, Multiframe has the option to store sections in the "Frames Group". The frames group is independent of the sections library and will be loaded each time you open the project.
Sections in the Frames Group can only be added/edited using Multiframe and cannot be accessed by Section Maker.



Note:

The correct library has to be loaded *before* opening the structural model. Else the members will loose their section properties and the following warning will appear:



It is best to close the model without saving, open the correct library and then re-open the model.

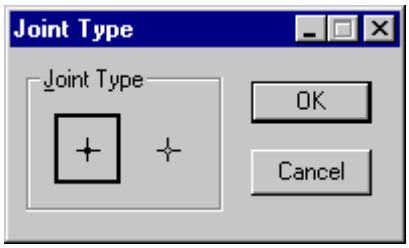
Joint Type

Multiframe allows you to use two basic types of joints: joints that are completely "rigid", i.e. they can transmit moments, or joints that are "pinned" i.e. they cannot transmit any moment.

To set the joint type

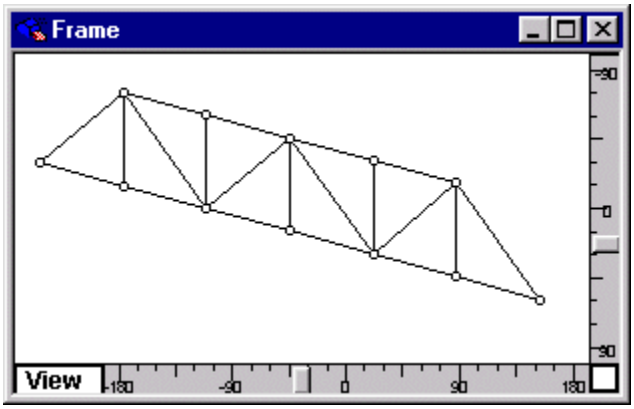
- **Select the joint or joints to be pinned**
- **Choose Joint Type from the Frame menu**

A dialog box will appear with icons indicating the two types of joints.



- **Click on the icon which represents the type of joint you require**
- **Click on the OK button**

The joints in the Frame window which are now pinned will be drawn with a circle on top of the joint to indicate that it is pinned.



Pinned joints are most useful in situations where all or most of the joints in a structure cannot support moment transfer. If you wish to model a situation where some of the members connected at a joint can transmit moment and some cannot, then you should use the Member Releases command explained below.

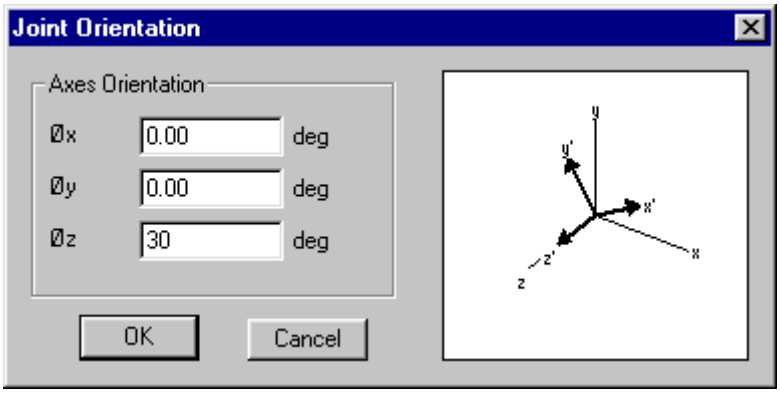
Joint Orientation

When modelling structures it is sometimes convenient to define restraints that are not aligned with the global coordinate system. In Multiframe, these types of restraints may be modelled by specifying the orientation of the local degrees of freedom at each joint. By default, these degrees of freedom are aligned with the global axes but this is not always necessary. The Joint Orientation dialog is used to specify an arbitrary local coordinate system at each joint. This local coordinate system defines the direction of the local degree of freedoms that are used for applying boundary conditions and loading to the structure.

The orientation of a joint is specified by 3 angles that measure the rotation of the local coordinate system from the global coordinate system. The right hand rule is used to determine the direction of the rotations.

To set the orientation of a joint;

- **Select the joint**
- **Choose "Joint Orientation…" from Frame menu.**
- **Enter the angles defining the orientation of the joint**
- **Press OK**



The orientation of a joint is displayed in the Frame and Load Windows. If the orientation of a joint is not aligned with the global coordinates then the axes of the local coordinates are displayed at the joint. The display of the local joint axes may be disabled via the Symbols dialog.

The joint displacements and joint reactions displayed in the Results Window are displayed in local joint coordinates. Global results for each of the displacement of reaction components are displayed in the tool tips in these tables.

Caution:

Multiframe files for structures using local joint loads are not compatible with versions of Multiframe prior to v7.5. These files may be read into the older version of Multiframe but analysis of these structures may cause the program to crash unexpectedly.

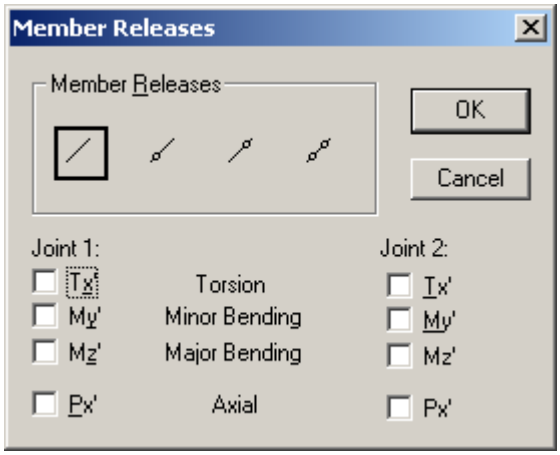
Member Releases

Multiframe allows you to use four basic types of members end releases. These are members that are pinned or rigid at one or both ends. If "rigid", they can transmit moments at the end, or if "pinned" they cannot transmit moments.

To set the member releases

- **Select the member or members to be pinned**
- **Choose Member Releases from the Frame menu**

A dialog box will appear with icons indicating the four types of member releases.



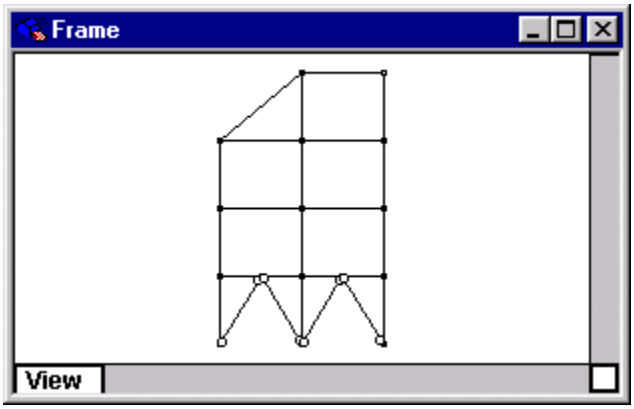
- **Click on the icon that represents the type of member, you require**

or

- **Click on the individual degrees of freedoms to select which are released.**
- **Click on the OK button**

When you specify the type of a member you have control over which degrees of freedom are released at the pinned ends of a member. You can release any of the three rotations at each end of a member. This helps prevent the problem of torsional mechanisms being created when a group of connected members all have the Tx' action released. To help prevent this, the default settings for a pinned end only release the major and minor rotations (My' and Mz'). You may also select to release the axial force at either end of the member. Shear releases are not available in Multiframe via this dialog. However they can be simulated using member end springs with no stiffness. These are described elsewhere in this manual.

The members in the Frame window with pinned ends will be drawn with a circle near the ends that are pinned. Although the pins are shown to be a small distance from the end of the member in the Frame window, for the purposes of analysis they are in fact infinitely close to the ends of the member. A pinned end releases moments about the local member axes.



Note that the assumed member releases are initially set to rigid/rigid and it is not necessary to explicitly define any releases on a member, unless you have previously set the member releases to rigid/pinned, pinned/rigid or pinned/pinned.

You can change the member releases on a member by double clicking on the pin icons at the ends of the member.

Member Types

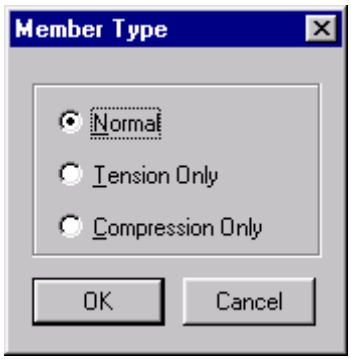
Multiframe allows you to specify a member as Tension only or Compression only. Tension only members will resist forces in the structure only when the member is in axial tension. Similarly, compression only members will resist forces in the structure only when the member is in axial compression. These types of members have been implemented in a general sense and changing a member to tension or compression only does not alter the member end releases. If a tension or compression only member is not required to resist moments then the appropriate member end releases should be applied.

Tension only and compression only members are taken into account by a nonlinear analysis. These types of members are removed or reinstated to the structure at the end of each iteration based upon the axial deformation of the member. The Tension only and compression only aspects of members are ignored in static linear, modal or time history analyses – they are treated as normal members in these cases.

To set the member type

- **Select the member or members to be modified**
- **Choose Member Type from the Frame menu**

A dialog box will appear with radio buttons indicating the types of members.

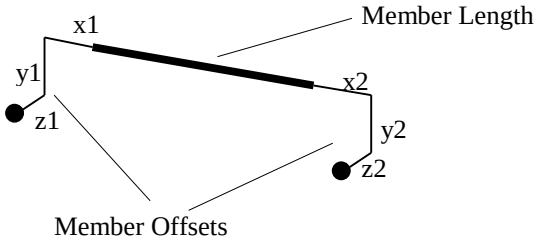


- **Click on the radio button corresponding to the type of member you require**
- **Click on the OK button**

Member Offsets

When modelling a structure there are situations in which members do not extend directly between two joints but are instead offset from the joints. In other situations, the intersection between two members may be relatively large and can be considered as a rigid link. In all these instances, the geometry of the model can be specified to more accurately model the structure by using rigid connections between the joints and the members. In Multiframe, these rigid connections are applied using Member Offsets.

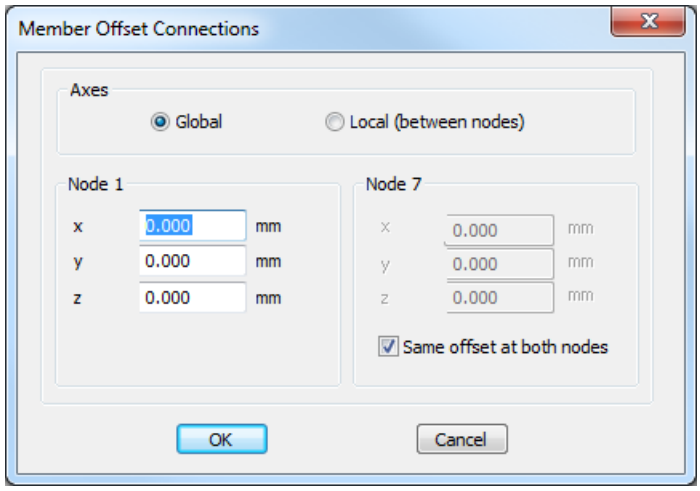
Member Offsets are rigid links between a joint and the end of a member. They are infinitely stiff and do not deform in bending, shear or axially. Loading on members with rigid offsets is applied to the flexible portion of the member only (see Member Length on the diagram below) and distances used to specify the position of loads are measured from the ends of the rigid offsets.



When Member Offsets are specified, Multiframe considers the length of the member to be the length of the flexible portion of the member. (Member Length in the diagram above)

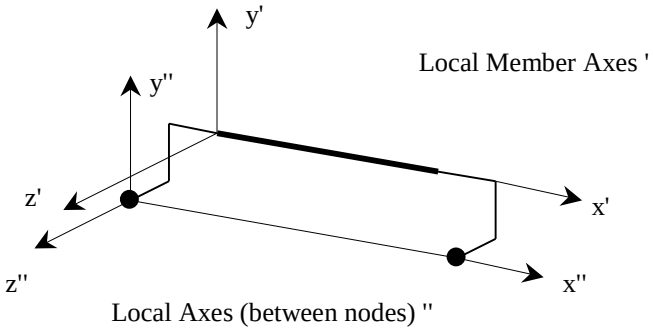
To set the Member Offsets on a member

- **Select the member or members to be modified**
- **Choose Member Offsets from the Frame menu**



- **Click on the radio button corresponding to the axes from which the offset connections will be measured.**
- **Type in the size of the offset connections at both ends of the members.**
- **Click on the OK button**

The size of the offset connections may be specified using global or local axes. If local axes are selected, the offset connections are measured relative to the local axes as defined by the joints at the end of the members. (see Local Axes between joints below) Note that this is distinctly different from the true local member axes which are aligned with the actual position and orientation of a member after the member offset has been applied.



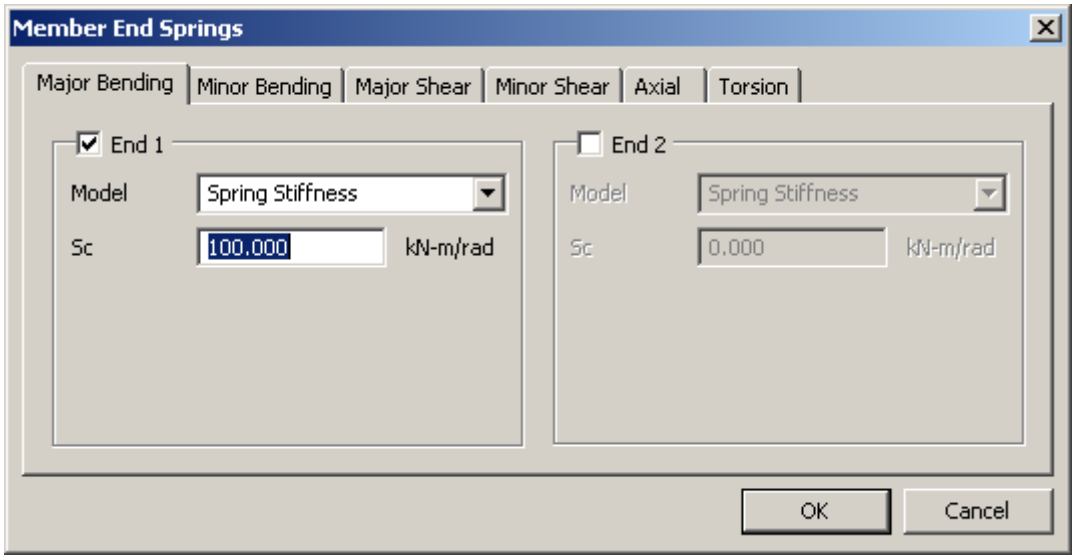
If the offset is the same along the length of the whole member, then tick the “Same offset at both nodes” checkbox and the offsets at the right hand node will be copied across.

Member End Springs

Modelling of semi-rigid connections in Multiframe is performed using Member end springs. Each member end spring is modelled as an individual item within the structural model. Furthermore each end spring is associated with only a single member and then only with one end of the member. This distinction is important as an end spring should not be considered to be properties of a member. Whenever a spring is specified at an end of a member a new end spring is added to the model.

To add Member or edit end springs to a member

- **Select the member or members to be modified**
- **Choose Member End Springs from the Frame menu**



For each component of the member end forces that you which to make semi rigid

- **Click on the tab corresponding to the component**
- **Click on End 1 and/or the End 2 check boxes to specify the ends of the member which are to be modelled using end springs.**

And for each end

- **Select the model used to determine the spring stiffness**
- **Specify the value of the parameter(s) used by the selected model**
- **Click on the OK button**

Three linear models are available for modelling the stiffness of the end springs. Each model defines a different method for specifying the stiffness of the spring. The models are as follows

1) Spring Stiffness (*Sc*) – The stiffness of the end spring is specified directly.

2) Rigidity Index (*r*) – The linear model of Lightfoot and LeMessurier (1974). This model computes the spring stiffness is direct proportion to the member stiffness according to the relationship.

$$Sc = r * 4EI/L$$

The value of *r* ranges from zero for a pinned or free connected to infinity for a rigid connection.

3) Fixity Factor (*n*) – Another linear model for spring stiffness was suggested by Romstad and Subramanian (1970) and Yu and Shanmugam (1986). The model uses the fixity factor (*n*) to define the spring stiffness using the following relationship

$$Sc = n/(1-n) * 4EI/L$$

The value o the fixity factor ranges from 0.0 to 1.0 where a value of 0.0 represents a pinned or free connection and value of 1.0 a rigid connection.

The last two methods were derived for modelling moment connections. However they have been generalised within Multiframe such that a similar scaling of the diagonal terms in the stiffness matrix is performed for shear, axial and torsional components of the end spring.

The properties of member end springs can be edited via the End Springs table in the Data Window. Member End Springs can be added and deleted via the Member Ends table in the Data Window. An end spring can be added by simply entering a number in the End Spring column, if this number is greater than the number of end springs in the model, then a new end spring is added using the default properties. If the number is that of an existing end spring then a new end spring is added using the properties of the existing spring. In both cases the number entered by the user will change to reflect he number of the new spring. An end spring is deleted from a member by simply setting the number of the end spring to zero.

Member Modelling

The Member Modelling dialog contains settings relevant to how a member is represented in a true physical model of the frame. At the moment the only setting contained in the dialog identifies if the member is a beam, column, brace or another component within the model.

To set the member modelling options for a member

- **Select the member or members to be modified**
- **Choose Member Modelling… from the Frame menu**

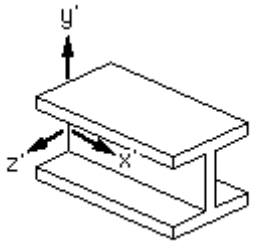
- **Select which component of the frame the member represents.**
- **Click on the OK button**

The component property of the member identifies what component within the frame that a member represents such as a column, primary beam or a brace. This information is used within the program help identify the most significant member at a connection. This is in turn used to automatically compute the cut-off distances at each ends of a member used in rendering of the frame.

Shear Area

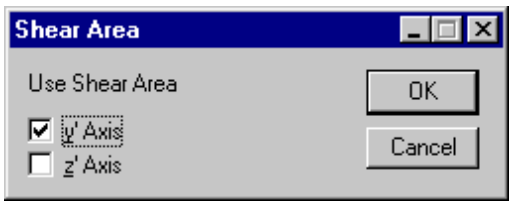
Multiframe allows shear area to be input and used in analysis. Using shear area can be switched on or off on a member-by-member basis.

To use shear area in a Multiframe analysis, a member must have a section type that has the shear area fields defined. Asx and Asy are the two fields in the Sections Library group that define the shear area for the sections local x and y axes, where x is the major axis. For all other sectional values, y is equivalent to local member axis y' and x is equivalent to local member axis z' in the negative direction.



The shear area values must be greater than zero, and the member must have the appropriate shear area flag set.

- **Choose Member Shear Area... from the Frame menu**



- **Select the local member axes for which shear area is to be used in analysis**

Remember Asx corresponds to the z' axis and Asy to the y' axis.

The members that use shear area in analysis are indicated in the last column in the Member Properties table in the Data window.

Applying Loads

Changes to the loads that are applied to the structure, may be made in the Load window. Loads may be added to the structure or removed from it and can be applied as a number of different load cases. These load cases may be factored together to produce combined load cases if desired.

Loads are entered in the current load units as controlled by the Units… command from the View Menu.

All of the commands under the Load menu operate on the current load case. The name of the current load case is displayed in the bottom left hand corner of the Load window. Also, the name of the current load case has a check mark beside it in the Case menu. You can set the current load case by choosing the appropriate name from the list of load case names under the Case menu or from the pop-up menu in the Load Case toolbar.

Loads are added by selecting joints or members and choosing the appropriate load command.

You can delete all the loads from joints or members by selecting the joints or members and pressing the Delete key.

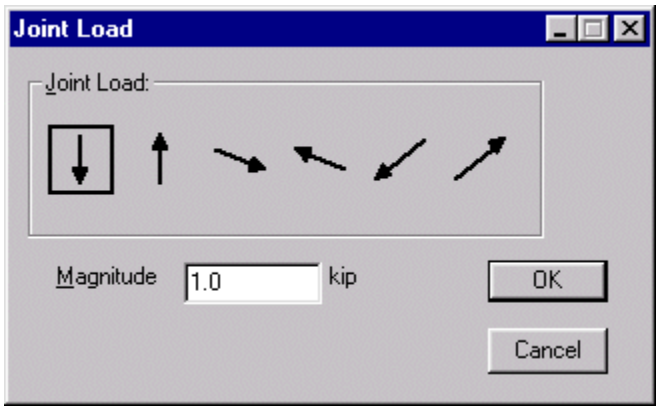
Joint Load

A joint load is a force that is applied at a joint in the structure. Joint loads can be applied relative to the global coordinate system in which case the forces act in a direction parallel to one of the reference x, y or z axes. Joint loads may also be specified using the local orientation of a joint in which case the loads act parallel to the direction of the local degrees of freedom at a joint.

To apply a load to a joint

- **Select the joint or joints to be loaded**
- **Choose Global Joint Load from the Load menu to apply a global load**
- **or**
- **Choose Local Joint Load from the Load menu to apply a local load**

A dialog box will appear with icons to indicate the direction of loading.



In a two dimensional view, there will be four icons indicating the four possible load directions. In the 3D view, all six possible icons will be displayed with the icons pointing in the direction of the appropriate axes in the current view.

- **Click on the icon which shows the direction in which the load is to act**
- **Type in a value for the magnitude of the load**
- **Click on the OK button**

There is no need to enter '+' or '-' signs for your load magnitudes. The directions are determined from the icon that you select.

If you wish to remove the joint loads from a joint, select the joint and choose Unload Joint from the Load menu. You can also double click on a joint to view a table of all the loads on the joint.

You can superimpose loads in the same direction on a single joint. Each time you use the Joint Load command, the new loads will be superimposed on any other joint loads you may have previously applied.

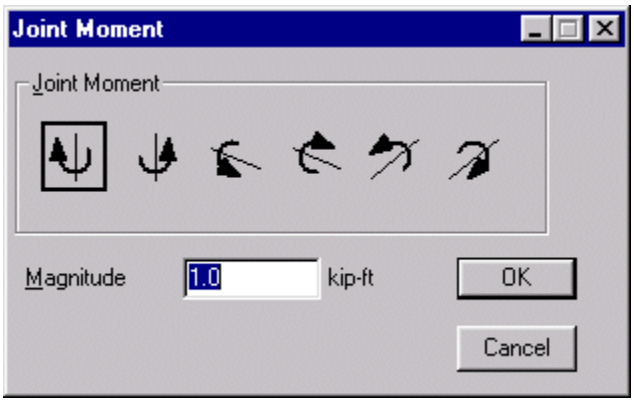
Joint Moment

A joint moment is a bending moment that is applied at a joint in the structure. Joint moments can be applied relative to the global coordinate system in which case the moments act about axes parallel to one of the global x, y or z axes. Joint moments may also be specified using the local orientation of a joint in which case the moments act about axes parallel to the direction of the local degrees of freedom at a joint.

To apply a moment to a joint

- **Select the joint or joints to be loaded**
- **Choose Global Joint Moment from the Load menu to apply a moment relative to the global axes**
- **or**
- **Choose Local Joint Moment from the Load menu to apply a moment relative to the nodal axes**

A dialog box will appear with icons to indicate the direction of loading.



In a two dimensional view, there will be four icons indicating the four possible moment directions. In the 3D view, all six possible icons will be displayed with the icons showing the direction of the action of the moments about the global axes in the current view. The line through the centre of each icon indicates the direction of the axis the moment is acting about.

- **Click on the icon which shows the direction in which the moment is to act**
- **Type in a value for the magnitude of the moment**
- **Click on the OK button**

There is no need to enter '+' or '-' signs for your moment magnitudes. The directions are determined from the icon that you select.

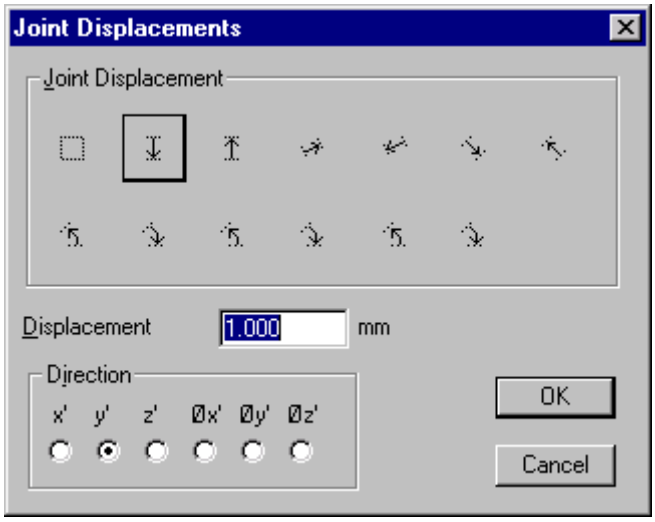
If you wish to remove the moments from a joint, select the joint and choose Unload Joint from the Load menu.

Prescribed Displacement

To prescribe a displacement at a joint

- **Select the joint or joints to undergo the prescribed displacement in the Load window**
- **Choose Prescribed Displacement from the Load menu**

A dialog box will appear with icons for the various prescribed displacements.



- **Click on the icon which shows displacement direction**
- **Type in a value for the displacement**
- **Click on the OK button**

Selecting the "No prescribed displacement" icon (the first icon in the list) will remove all prescribed displacements from the selected joints.

Prescribed displacements act in the local coordinate system of the joint to which they are applied. Each of the icons represents a direction for the displacement relative to one of the joint's reference axes. As you click on the icons that indicate the various displacements, the directions of each displacement will be indicated by the radio buttons at the bottom of the dialog. You can also click on the radio buttons to choose which direction you wish the displacement to act in.

A prescribed joint displacement affects the current load case only.

The properties of a prescribed displacement may be edited by double clicking on the icon representing the prescribed displacement.

Global Distributed Load

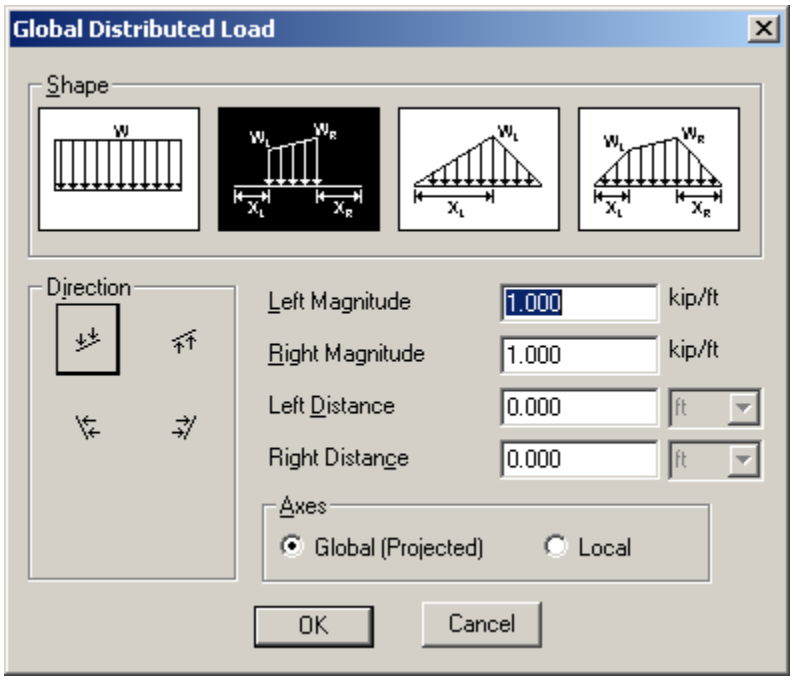
Multiframe allows loading on members to be applied relative to the direction of the global coordinate system or relative to the direction of the local member coordinate system. Loads that are applied at an angle to either of these systems, can be modelled by using vector components of the loads.

A global distributed load is a load which is distributed along all or part of a member, and acts in a direction parallel to one of the reference x, y or z axes.

To apply a global distributed load to a member

- **Select the member or members to be loaded**
- **Choose Global Dist'd Load from the Load menu or short cut menu**

A dialog box will appear with icons to indicate the shape and direction of loading.



- **Click on the icon which shows the shape of the load**

In a two dimensional view, there will be four icons indicating the four possible loading directions. In the 3D view, all six possible icons will be displayed with the patterns in the icons showing the direction of the action of the loads. Each icon has a pattern with lines running parallel to the direction of action.

- **Click on the icon which shows the direction in which the load is to act**
- **Type in values for the magnitude of the load at each end**
- **Click the axes in which to apply the load**

The axis of the load determines how the load is distributed along the members. With Global or Projected axes, the magnitude of a global distributed load refers to its load per length where the length is measured perpendicular to the direction of the load. This means a vertical distributed load applied to an inclined member will apply a total load equivalent to the magnitude of the load times the horizontal projected length of the member. For loads applied in local axes, the magnitude of the distributed load refers to its load per length where the length is the actual length of the load measured along the member.

- **Click on the OK button**

Note that when working in dialog boxes, you can use the Tab key to move from one editing box to the next. If you do this in this case, the right hand end magnitude will automatically be set to the same value as the left hand end magnitude. You may type over this second value if you wish to have a non-uniform load.

There is no need to enter '+' or '-' signs for your load magnitudes. The directions are determined from the icon that you select.

When you enter positions of loads in the member loading dialogs or the Data window, you can enter calculation expressions for the position. For example, if you want a load to start at one third span you can enter L/3 for its start position. If you enter this in a load dialog, this expression will be calculated for all the selected members. This means you can apply this load to a number of members of different lengths simultaneously. You can also enter more complicated expressions such as 2*L/3 or 1.35+(L-4)/2. The variable L is always available and contains the length of the member, the syntax of the expressions is the same as that used in the CalcSheet.

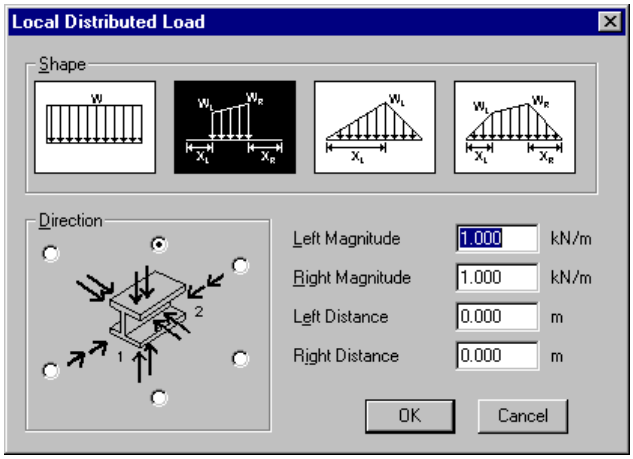
Local Distributed Load

A local distributed load is a load which is distributed along all or part of the member and acts in a direction either normal (shear) or tangential (axial) to the member.

To apply a local distributed load to a member

- **Select the member or members to be loaded**
- **Choose Local Dist'd Load from the Load menu or short cut menu**

A dialog box will appear with icons to indicate the shape of the load and its direction relative to the member.



- **Click on the icon which shows the shape of the load**
- **Click on the icon which shows the direction in which the load is to act**
- **Type in values for the magnitude of the load at each end**
- **Click on the OK button**

There is no need to enter '+' or '-' signs for your load magnitudes. The directions are determined from the icon that you select.

When you enter positions of loads in the member loading dialogs or the Data window, you can enter calculation expressions for the position. For example, if you want a load to start at one third span you can enter L/3 for its start position. If you enter this in a load dialog, this expression will be calculated for all the selected members. This means you can apply this load to a number of members of different lengths simultaneously. You can also enter more complicated expressions such as 2*L/3 or 1.35+(L-4)/2. The variable L is always available and contains the length of the member, the syntax of the expressions is the same as that used in the CalcSheet.

Global Panel Load

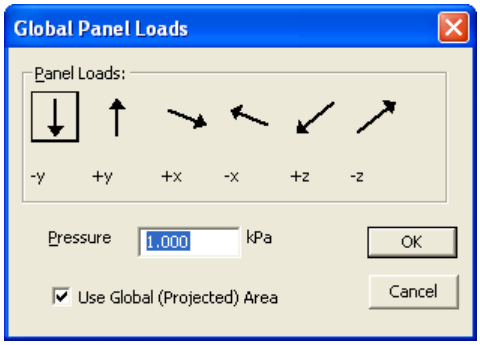
Multiframe allows loading on panels to be applied relative to the direction of the global coordinate system or relative to the direction of the local panel coordinate system.

A global panel load is a load which is uniformly distributed along all areas of a load panel, and acts in a direction parallel to one of the reference x, y or z axes.

To apply a global distributed load to a panel

- **Select the load panel or panels to be loaded**
- **Choose Global Panel Load from the Load menu or short cut menu**

A dialog box will appear with icons to indicate the direction of loading.



- **Click on the icon which shows the load direction**

In a two dimensional view, there will be four icons indicating the four possible load directions. In the 3D view, all six possible icons will be displayed with the icons pointing in the direction of the appropriate axes in the current view.

- **Click on the icon which shows the direction in which the load is to act**
- **Type in a value for the pressure**
- **Click on the OK button**

There is no need to enter '+' or '-' signs for your load values. The directions are determined from the icon that you select.

If you wish to remove the panel loads from a panel, select the panel and choose Unload panel from the Load menu. You can also double click on a panel to view a table of all the loads on the panel.

The axis of the load determines how the load is projected in the panel areas. With Global axes, the magnitude of a global panel load refers to its load per square meters where the area is measured perpendicular to the direction of the load. This means a vertical distributed load applied to an inclined panel will apply a total load equivalent to the magnitude of the load times the horizontal projected area of the panel. For loads applied in local axes, the magnitude of the distributed load refers to its load per square meters where the area is the actual area of the load measured along the panel plane.

- **Click on the OK button**

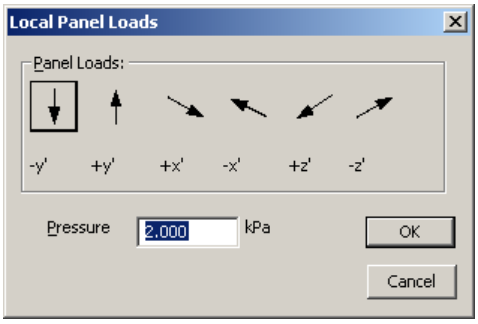
Local Panel Load

A local panel load is a load which is distributed along full area of the panel and acts in a direction either normal (shear) or tangential (friction) to the panel.

To apply a local distributed load to a panel

- **Select the load panel or panels to be loaded**
- **Choose Local Panel Load from the Load menu or short cut menu**

A dialog box will appear with icons to indicate the direction of loading.



- **Click on the icon which shows the load direction**

In a two dimensional view, there will be four icons indicating the four possible load directions. In the 3D view, all six possible icons will be displayed with the icons pointing in the direction of the appropriate local axes of the panel in the current view.

- **Click on the icon which shows the direction in which the load is to act**
- **Type in a value for the pressure of panel loading**
- **Click on the OK button**

There is no need to enter '+' or '-' signs for your load magnitudes. The directions are determined from the icon that you select.

If you wish to remove the panel loads from a panel, select the panel and choose Unload panel from the Load menu. You can also double click on a panel to view a table of all the loads on the panel.

Global Patch Distributed Load

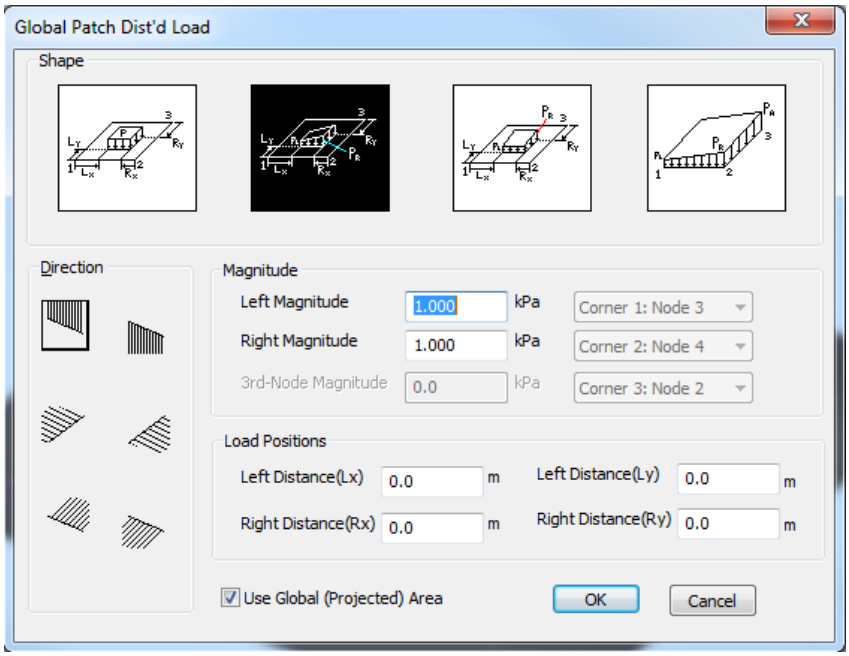
Multiframe allows loading on patches to be applied relative to the direction of the global coordinate system or relative to the direction of the local patch coordinate system. Loads can be applied either constantly or linearly.

A global distributed load is a load which is distributed along all or part of a patch, and acts in a direction parallel to one of the reference x, y or z axes.

To apply a global distributed load to a patch

- **Select the patch or patches to be loaded**
- **Choose Global Patch Dist'd Load from the Load menu or short cut menu**

A dialog box will appear with icons to indicate the shape and direction of loading.



- **Click on the icon which shows the shape of the load**

In a two dimensional view, there will be four icons indicating the four possible loading directions. In the 3D view, all six possible icons will be displayed with the patterns in the icons showing the direction of the action of the loads. Each icon has a pattern with lines running parallel to the direction of action.

- **Click on the icon which shows the direction in which the load is to act**
- **Type in values for the magnitude of the load at each end**
- **Select global or projected area in which to apply the load**

The axis of the load determines how the load is distributed along the patches. With Global or Projected areas, the magnitude of a global distributed load refers to its load per square unit where the area is measured perpendicular to the direction of the load. This means a vertical distributed load applied to an inclined patch will apply a total load equivalent to the magnitude of the load times the horizontal projected area of the patch. For loads applied in patch local area, the magnitude of the distributed load refers to its load per square unit where the unit is the actual area of the load measured along the patch.

- **Click on the OK button**

Note that when working in dialog boxes, you can use the Tab key to move from one editing box to the next. There is no need to enter '+' or '-' signs for your load magnitudes. The directions are determined from the icon that you select.

When you enter positions of loads in the patch loading dialogs or the Data window, you can enter calculation expressions for the position. For example, if you want a load to start at one third patch sides you can enter L/3 for the left distances (Lx and/or Ly) of start position. If you enter this in a load dialog, this expression will be calculated for all the selected patches. This means you can apply this load to a number of patches of different dimensions simultaneously. You can also enter more complicated expressions such as 2*L/3 or 1.35+(L-4)/2. The variable L is always available and contains the length of the patch side, the syntax of the expressions is the same as that used in the CalcSheet.

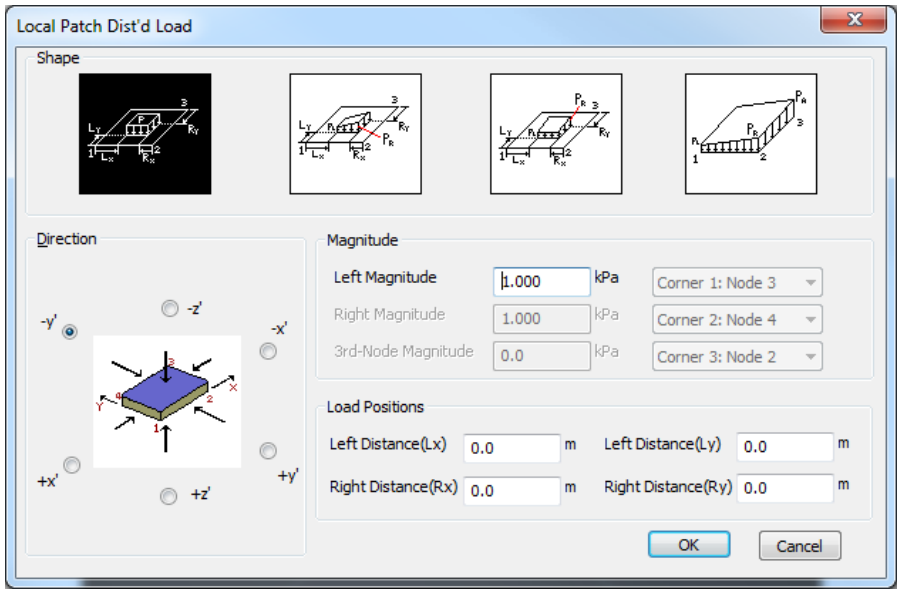
Local Patch Distributed Load

A local distributed load is a load which is distributed along all or part of the patch and acts in a direction either normal (shear) or tangential (axial) to the patch.

To apply a local distributed load to a patch

- **Select the patch or patches to be loaded**
- **Choose Local Dist'd Load from the Load menu or short cut menu**

A dialog box will appear with icons to indicate the shape of the load and its direction relative to the patch.



- **Click on the icon which shows the shape of the load**
- **Click on the icon which shows the direction in which the load is to act**
- **Type in values for the magnitude of the load at each end**
- **Click on the OK button**

Note that when working in dialog boxes, you can use the Tab key to move from one editing box to the next. There is no need to enter '+' or '-' signs for your load magnitudes. The directions are determined from the icon that you select

When you enter positions of loads in the patch loading dialogs or the Data window, you can enter calculation expressions for the position. For example, if you want a load to start at one third patch sides you can enter L/3 for the left distances (Lx and/or Ly) of start position. If you enter this in a load dialog, this expression will be calculated for all the selected patches. This means you can apply this load to a number of patches of different dimensions simultaneously. You can also enter more complicated expressions such as 2*L/3 or 1.35+(L-4)/2. The variable L is always available and contains the length of the patch side, the syntax of the expressions is the same as that used in the CalcSheet.

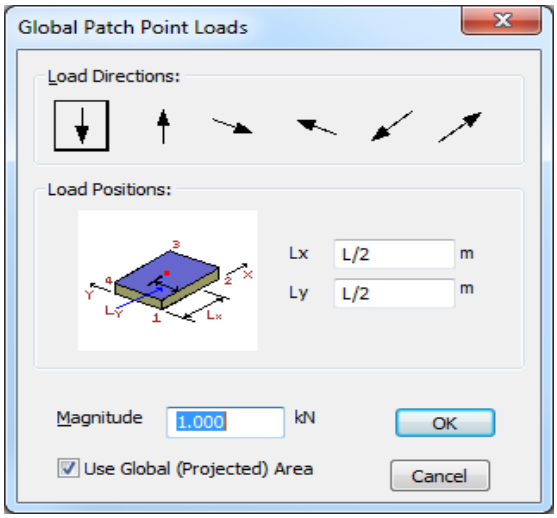
Global Patch Point Load

A global patch point load is a concentrated load that acts at a position within a patch boundary and acts in a direction parallel to one of the reference x, y or z axes.

To apply a global point load to a patch

- **Select the patch or patches to be loaded**
- **Choose Global Patch Point Load from the Load menu or short cut menu**

A dialog box will appear with icons to indicate the direction of loading.



In a two dimensional view, there will be four icons indicating the four possible load directions. In the 3D view, all six possible icons will be displayed with the icons pointing in the direction of the appropriate axes in the current view.

- **Click on the icon which shows the direction in which the load is to act**
- **Enter the position of the load**
- **Type in the value for the magnitude of the load**
- **Click on the OK button**

There is no need to enter '+' or '-' signs for your load values. The directions are determined from the icon that you select.

When entering positions of loads in the Patch Point Loading dialog, it is possible to enter calculation expressions . For example, to place a load at the centre of the patch enter L/2 for the left distance and L/2 for the up distance. It is possible to apply this loads in this way to a number of patches with different simultaneously. The individual value of L for each patch is used to calculate the position of each load.

To remove the patch loads from a patch, select the patch and choose Unload patch from the Load menu. You can also double click on a patch to view a table of all the loads on the patch.

Only one point load is allowed to be added to each patch per load case.

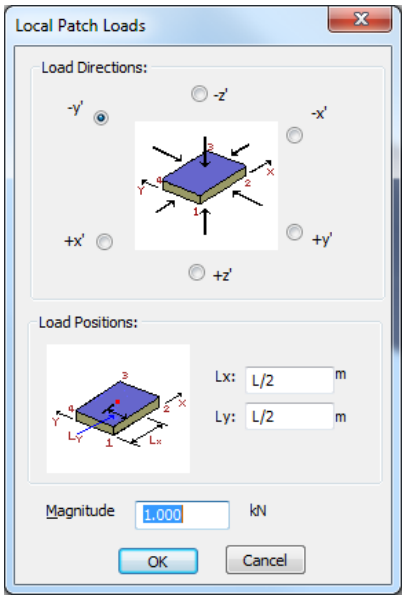
Local Patch Point Load

A local patch point load is a concentrated load that acts at a position within a patch boundary and acts in a direction parallel to one of the reference x, y or z axes.

To apply a local point load to a patch

- **Select the patch or patches to be loaded**
- **Choose Local Patch Point Load from the Load menu or short cut menu**

A dialog box will appear with icons to indicate the direction of loading.



All six possible radio buttons will be displayed pointing in the direction of the local axes in the current view.

- **Click on the radio button which shows the direction in which the load is to act**
- **Enter the position of the load**
- **Type in the value for the magnitude of the load**
- **Click on the OK button**

There is no need to enter '+' or '-' signs for your load values. The directions are determined from the radio button that you select.

When entering positions of loads in the Patch Loading dialog, it is possible to enter calculation expressions . For example, to place a load at the centre of the patch enter L/2 for the left distance and L/2 for the up distance. It is possible to apply this loads in this way to a number of patches with different simultaneously. The individual value of L for each patch is used to calculate the position of each load.

If you wish to remove the patch loads from a patch, select the patch and choose Unload patch from the Load menu. You can also double click on a patch to view a table of all the loads on the patch.

Only one point load is allowed to be added to each patch per load case.

Patch Edge Loads

Multiframe allows edge loading on patches to be applied relative to the direction of the local patch coordinate system.

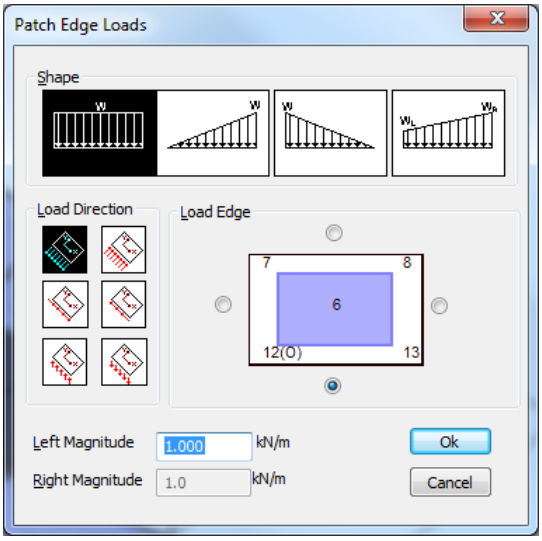
A local patch edge load is a load which is distributed along the edges the patch and acts in a direction either normal or tangential to the patch edge.

To apply a local edge distributed load to a patch

- **Select the patch or patches to be loaded**
- **Choose Local Patch Edge Load from the Load menu or short cut menu**

A dialog box will appear with icons to indicate the shape of the load and its direction relative to the patch edge.

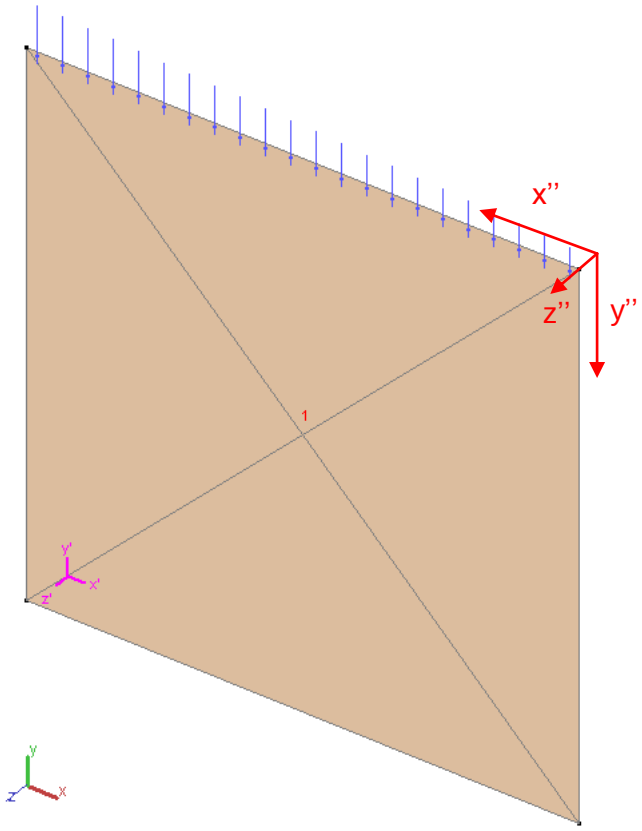
All six icons will be displayed in the direction of the appropriate local axes of the patch in the current view.



- **Click on the icon which shows the shape of the load**
- **Click on the icon which shows the direction in which the load is to act**
- **Click on the patch edge which the load is applied to**
- **Type in a value for the magnitude of the load at its end**
- **Click on the OK button**

There is no need to enter '+' or '-' signs for your load magnitudes. The directions are determined from the icon that you select.

The axis convention used for the direction of edge loads is shown in the diagram below. Edge loads along the direction of the patch edge are in the local x'' direction, edge loads parallel with the patch are in the y'' direction and edge loads normal to the patch are in the z'' direction. This sign convention is used in the data tables.



If you wish to remove the patch edge loads from a patch, select the patch and choose Unload patch from the Load menu. You can also double click on a patch to view a table of all the loads on the patch.

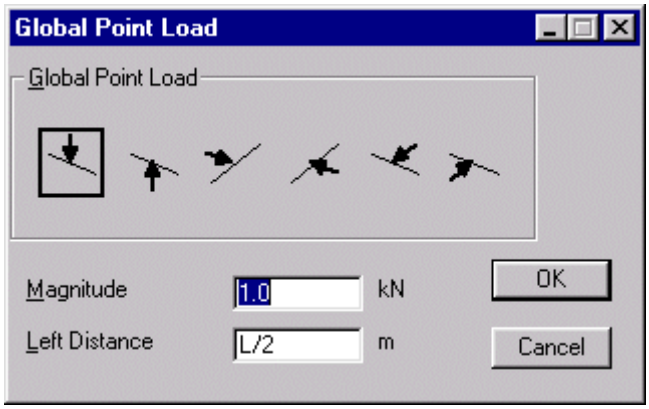
Global Point Load

A global point load is a concentrated load that acts at a position part way along a member and acts in a direction parallel to one of the reference x, y or z axes.

To apply a global point load to a member

- **Select the member or members to be loaded**
- **Choose Global Point Load from the Load menu or the short cut menu**

A dialog box will appear with icons to indicate the direction of loading.



In a two dimensional view, there will be four icons indicating the four possible loading directions. In the 3D view, all six possible icons will be displayed with the direction of the arrows in the icons showing the direction of the action of the loads as they will be displayed in the current view.

- **Click on the icon which shows the direction in which the load is to act**
- **Type in the value for the magnitude of the load**
- **Press Tab and type in the position of the load measured from joint 1**
- **Click on the OK button**

There is no need to enter '+' or '-' signs for your load magnitudes. The directions are determined from the icon that you select.

When you enter positions of loads in the member loading dialogs or the Data window, you can enter calculation expressions for the position. For example, if you want a load to be at mid span you can enter L/2 for its position. If you enter this in a load dialog, this expression will be calculated for all the selected members. This means you can apply this load to a number of members of different lengths simultaneously. You can also enter more complicated expressions such as 2*L/3 or 1.35+(L-4)/2. The variable L is always available and contains the length of the member, the syntax of the expressions is the same as that used in the CalcSheet.

Multiple loads of the same magnitude and direction may be added to the members by entering a comma separated list of load positions.

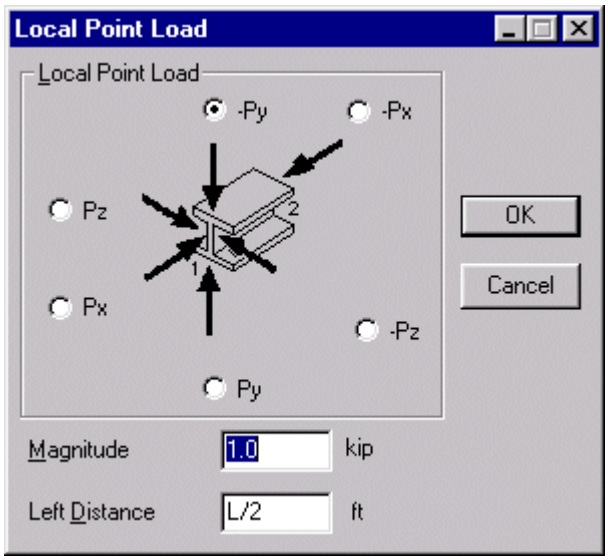
Local Point Load

A local point load is a concentrated load that acts part way along a member and acts in a direction normal (shear) or parallel (axial) to the member.

To apply a local point load to a member

- **Select the member or members to be loaded**
- **Choose Local Point Load from the Load menu or the short cut menu**

A dialog box will appear with icons to indicate the direction of loading relative to the member.



- **Click on the icon which shows the direction in which the load is to act**
- **Type in the value for the magnitude of the load**
- **Press Tab and type in the position of the load measured from joint 1**
- **Click on the OK button**

When you enter positions of loads in the member loading dialogs or the Data window, you can enter calculation expressions for the position. For example, if you want a load to be at mid span you can enter L/2 for its position. If you enter this in a load dialog, this expression will be calculated for all the selected members. This means you can apply this load to a number of members of different lengths simultaneously. You can also enter more complicated expressions such as 2*L/3 or 1.35+(L-4)/2. The variable L is always available and contains the length of the member, the syntax of the expressions is the same as that used in the CalcSheet.

Multiple loads of the same magnitude and direction may be added to the members by entering a comma separated list of load positions.

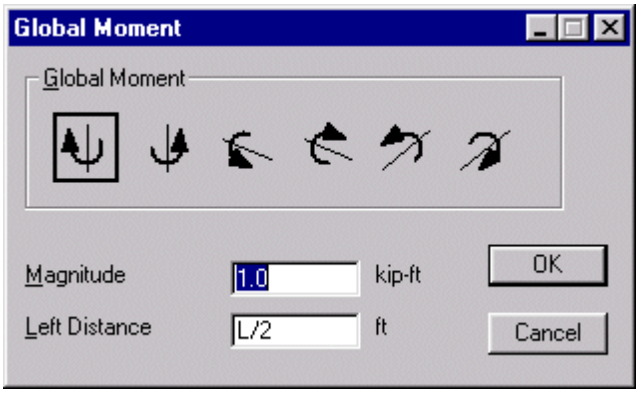
Global Moment

A global moment is a bending moment that acts part way along a member and acts about one of the reference x, y or z axes.

To apply a global moment to a member

- **Select the member or members to be loaded**
- **Choose Global Moment from the Load menu or the short cut menu**

A dialog box will appear with icons to indicate the direction of the moment.



In a two dimensional view, there will be two icons indicating the two possible moments which can be applied. In the 3D view, all six possible icons will be displayed with the direction of the arrows in the icons showing the direction of the action of the loads, as they will be displayed in the current view.

- **Click on the icon which shows the direction in which the moment is to act**
- **Type in the value for the magnitude of the moment**
- **Press Tab and type in the position of the load measured from joint 1**
- **Click on the OK button**

There is no need to enter '+' or '-' signs for your moment magnitudes. The directions are determined from the icon that you select.

When you enter positions of loads in the member loading dialogs or the Data window, you can enter calculation expressions for the position. For example, if you want a load to be at mid span you can enter L/2 for its position. If you enter this in a load dialog, this expression will be calculated for all the selected members. This means you can apply this load to a number of members of different lengths simultaneously. You can also enter more complicated expressions such as 2*L/3 or 1.35+(L-4)/2. The variable L is always available and contains the length of the member, the syntax of the expressions is the same as that used in the CalcSheet.

Multiple loads of the same magnitude and direction may be added to the members by entering a comma separated list of load positions.

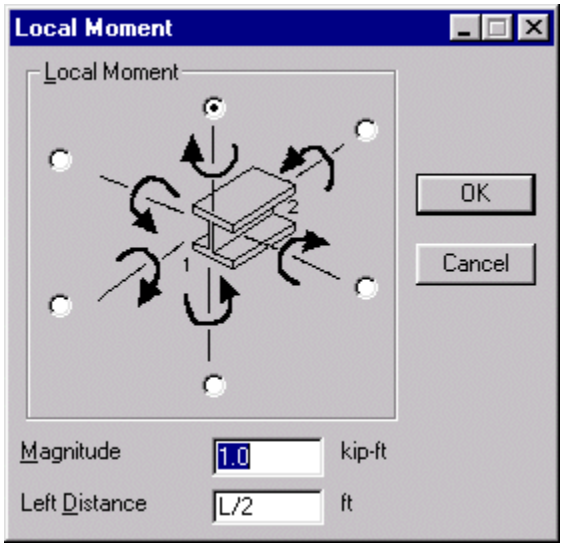
Local Moment

A local moment is a bending moment that acts part way along a member and acts about one of the local axes.

To apply a local moment to a member

- **Select the member or members to be loaded**
- **Choose Local Moment from the Load menu or the short cut menu**

A dialog box will appear with icons to indicate the direction of the moments relative to the member.



- **Click on the icon which shows the direction in which the moment is to act**
- **Type in the value for the magnitude of the moment**
- **Press Tab and type in the position of the moment measured from joint 1**
- **Click on the OK button**

When you enter positions of loads in the member loading dialogs or the Data window, you can enter calculation expressions for the position. For example, if you want a load to be at mid span you can enter L/2 for its position. If you enter this in a load dialog, this expression will be calculated for all the selected members. This means you can apply this load to a number of members of different lengths simultaneously. You can also enter more complicated expressions such as 2*L/3 or 1.35+(L-4)/2. The variable L is always available and contains the length of the member, the syntax of the expressions is the same as that used in the CalcSheet.

Multiple loads of the same magnitude and direction may be added to the members by entering a comma separated list of load positions.

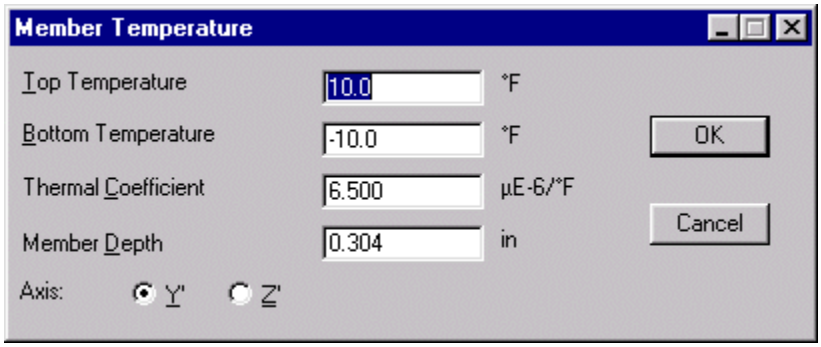
Thermal Load

A thermal load is a load resulting from a temperature differential in the structure between a member and the ambient temperature. A thermal load may also result from a temperature gradient through the depth of a member.

To apply a thermal load to a member

- **Select the member or members to be loaded**
- **Choose Thermal Load from the Load menu or the short cut menu**

A dialog box will appear with fields for the magnitude and depth of the thermal load.



- **Choose whether the temperature varies through the depth (web or y' axis direction) or the breadth (flange or z' axis direction) of the member**
- **Type in the temperatures at the top and bottom of the member**

These two temperatures will be the same if the member is at a constant temperature. If the load is applied in the z' direction top refers to the positive z' side while bottom refers to the negative z' side.

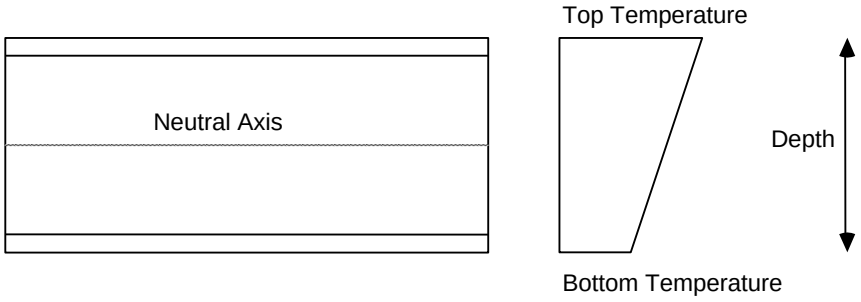
- **Type in a value for the thermal coefficient for the member material**

The thermal coefficient is entered in units of microstrain per degree.

- **Type in a value for the depth of the thermal load**
- **Click on the OK button**

All the temperature values are variations in degrees from the ambient temperature. For example, a bridge deck with the top of the beams at 45° and the bottom of the beams at an ambient of 20° would have top and bottom temperatures of 25° and 0° respectively.

The thermal gradient is assumed to be symmetric about the neutral axis of the member. That is, the top temperature occurs at half the depth above the neutral axis and varies linearly down to a point half the depth below the neutral axis. Note that the depth of the load does not have to be the same as the depth of the member.



Editing Loads Numerically

You can edit the values and positions of loads and prescribed displacements numerically in tables of data that may be displayed in the Data window. These tables display Joint Loads, Prescribed Displacements, Member Loads and Thermal Loads. You can choose which table to display by choosing the appropriate item from the Data sub-menu under the Display menu.

To change the value or position of a load

- **Click on the number to be changed**
- **Type in the new value for the load**

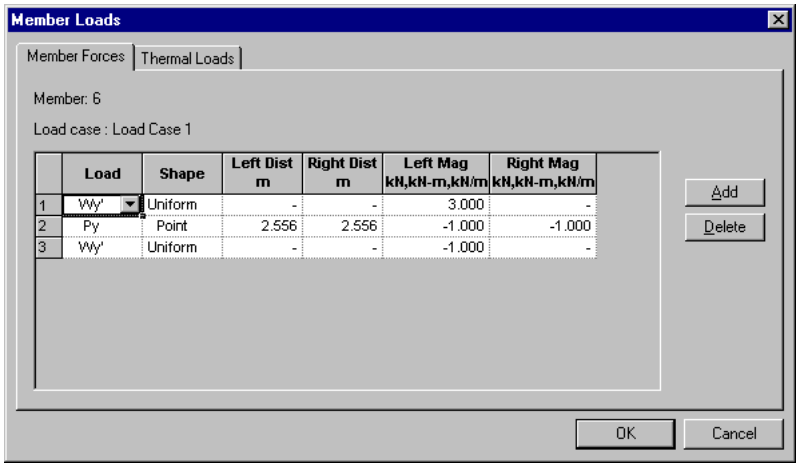
- **Press the enter key**

If you change the position or direction of a load, the drawing in the Load window will be updated to reflect the change.

Note that the loads for the current load case only will be displayed in the Data window. You cannot change loads for a combined load case, a Time History load case or a self weight load case.

Load Properties

You can also edit loads numerically by double clicking on a member in the Load window. This will display a dialog with a table of member loads and a table of thermal loads.



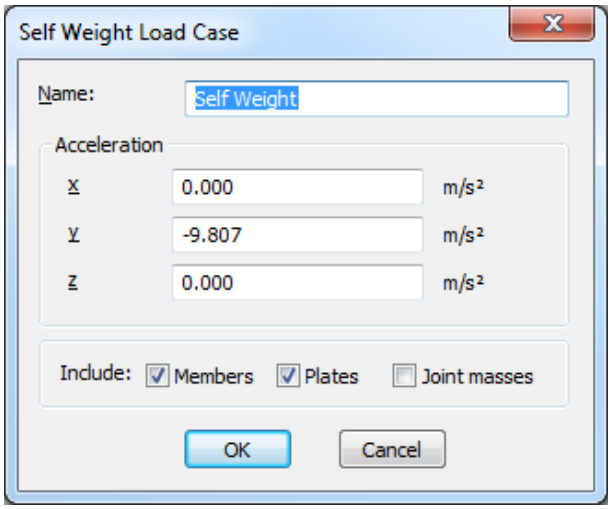
All of these values can be edited in this table and you can also add and delete loads using the buttons in the dialog. Loads on joints can be changed in a similar way by double clicking on the joint.

Self Weight

Multiframe allows you to automatically include the self weight of the structure as a separate load case.

- **Choose Self Weight… from the Load menu or the short cut menu**

A dialog will be displayed which allows you to enter the acceleration to be applied to the mass of the structure to convert it into a load.



Acceleration

You will usually use the default setting which applies a standard gravity acceleration to the structure. However if you are investigating inertial or seismic effects, you may wish to apply accelerations in the x or z directions.

Include

For versions of Multiframe 14 and onwards, it is possible to include Joint Masses in the self weight calculation. By default they are turned off and are off for any models created in older versions of Multiframe.

Similarly it is possible to include or exclude any combination of Members, Plates and Joint Masses.

After clicking OK, a new load case named Self Weight will be created. You can create any number of self weight load cases with different acceleration factors. You should make sure that you have set all of the section types for the structure before you use the self weight command. If any of the sections used in the structure have zero mass, Multiframe will prompt you with a dialog to go ahead with or cancel the use of self weight. If you wish to use self weight with custom sections, make sure you enter an appropriate value for the mass when adding the section to the library.

Member Self Weight

If you add a self weight case, you can specify whether or not an individual member's weight is to be included by using the Member Self Weight command from the Load menu.

To specify whether a member's weight should be included in self weight

- **Make sure a self weight load case is the current load case**
- **Select the member(s) to be changed in the Load window**
- **Choose Member Self Weight from the Load menu**

- **Click the Ignore or Include radio button as appropriate**
- **Click OK**

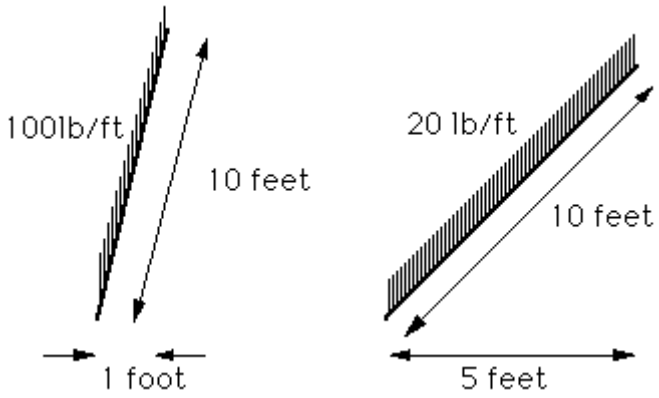
If you have more than one self weight load case, you can turn on the check box to have the same self weight settings used for all self weight cases, otherwise the settings may be different for each self weight load case.

The Member Self Weight command only affects the self weight for static analysis. If you are doing a dynamic analysis in 4D, the member's weight is controlled by the Member Masses command from the Frame menu. The default setting is that the mass is included.

A self weight load case can be combined with other load cases in the usual way. The numerical values of the self weight loads are not displayed in the Data window. These values will be updated just prior to analysis to ensure that the weight for any members you have changed are included correctly.

Self Weight Loads

The loads for self weight are included in two ways. Members which are vertical are loaded with an axially distributed load, Wz', acting in the vertical direction, from joint 2 to joint 1. You will see values for this load in printouts or in factored load cases that are made up from a self weight load case. Members that are not vertical, are loaded with a vertical distributed load scaled to take into account the horizontal projected length of the member (see Global Distributed Load above). Because global distributed loads are applied in units of force/horizontal length, the magnitude of the distributed load must be scaled to take into account.



For example, a 10ft long member weighing 100 pounds with a horizontal projected length of 1 foot would require a distributed load of 100lb/ft to apply the correct self weight. The same 10 foot long member with projected length of 5 feet would only require a distributed load of 20lb/ft to correctly apply the self weight. For this reason, if you have members that are very nearly vertical and therefore have a very small projected length, you will see very high values for the distributed load applied due to self weight.

Load Cases

Multiframe allows you to create a number of different types of load cases to manage different loading conditions

Multiple Load Cases

You can apply up to 500 load cases to a structure at a time. You may define some of these load cases to be factored combinations of existing load cases. If you make a change to a load case that is used by other load cases, the factored load cases will automatically be updated to reflect the change. You can automatically construct a load case that includes the self weight of the structure as outlined in the previous section.

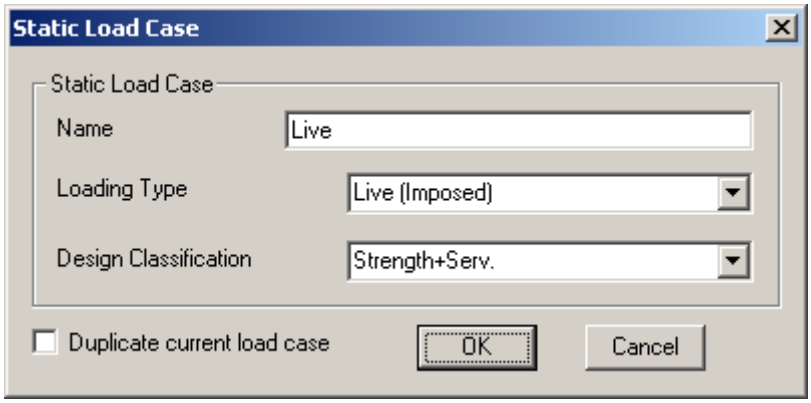
Initially, one load case is defined titled 'Load Case 1'. You can edit one load case at one time. The current load case is indicated by a check mark on the appropriate item in the lower part of the Case menu and is shown in the lower left hand corner of the Load window and the Plot windows when two dimensional views are displayed. If the load case is a factored combination of other cases, it will have (Combined) displayed after the name of the case in the Load window. You cannot edit or remove loads from a factored load case. If you change loads in a load case that is used by a combined load case, the combined case will be automatically updated.

Note that a factored load case can be a factored combination of a factored load case, but cannot include a factored combination of a load case higher in the load case list. For example, load case 3 can include combinations of load cases 1 and 2 but could not include load case 4.

Adding a Static Load Case

To add a new load case

- **Choose Add Case from the Case menu**
- **Select Static from the Add Case sub-menu.**



- **Type in a name for the load case.**

The name you give to the load case must not contain any of the following characters; ^ ! < / (

- **Select a Loading Type and the Design classification.**

This information will be used in future when doing design using Steel Designer.

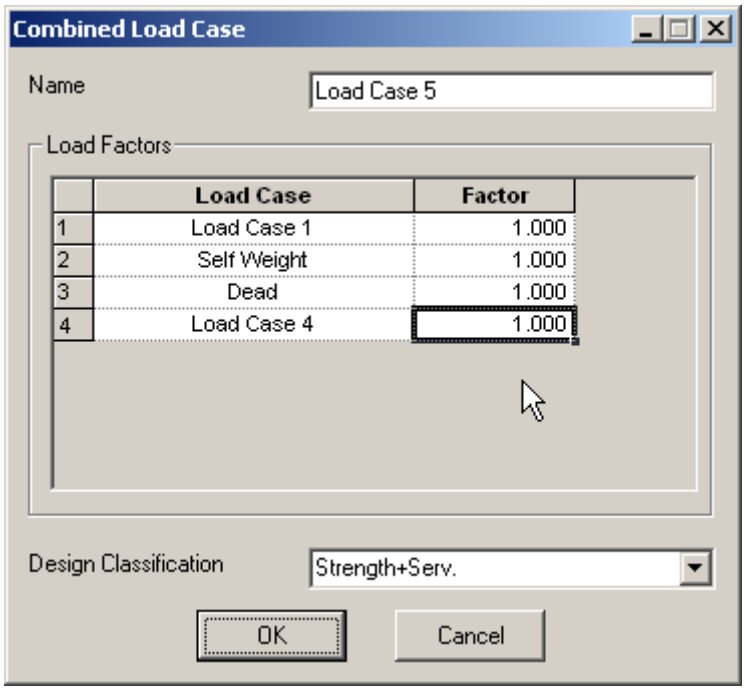
- **Click on the OK button**

If you turn on the "Duplicate current load case" check box, the new load case will initially contain the loads in the current load case. This is useful if you have two load cases that are very similar, and one can be made from a small variation on the other.

Adding a Factored Load Case

To add a new load case

- **Choose Add Case from the Case menu**
- **Select Static Combined from the Add Case sub-menu.**



- **Type in a name for the load case (or leave the default name if you wish)**

The name you give to the load case must not contain any of the following characters; ^ ! < / (

- **Enter the load factors for the existing load cases**
- **Select Design classification.**

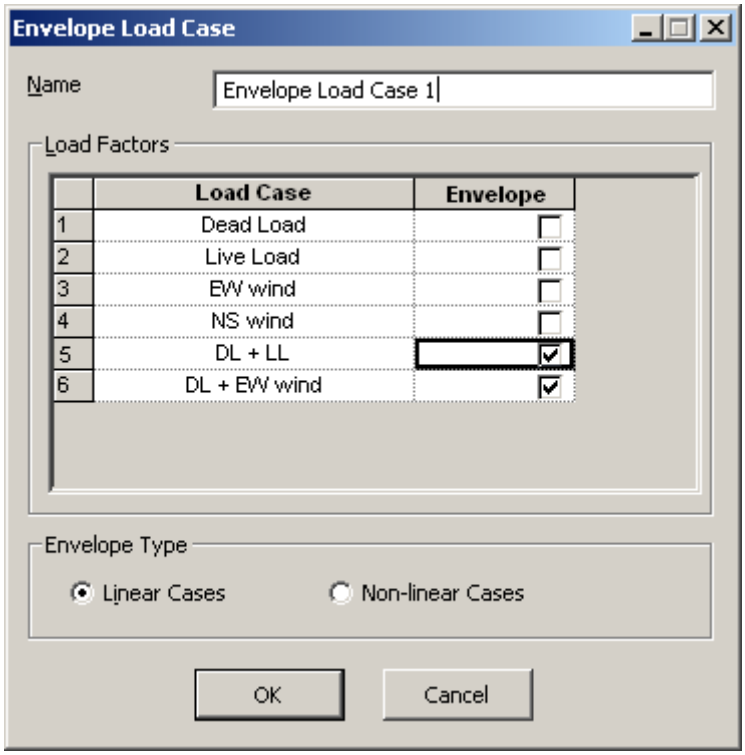
This information will be used in future when doing design using Steel Designer.

- **Click on the OK button**

Adding a Envelope Load Case

To add a new load case

- **Choose Add Case from the Case menu**
- **Select Envelope from the Add Case sub-menu.**



- **Type in a name for the load case (or leave the default name if you wish)**
- **Select load cases the load cases to be enveloped**
- **Select the results to be enveloped, either Linear or Nonlinear.**
- **Click on the OK button**

Editing a Load Case

To change the name or load factors of a load case

- **Choose Edit Case from the Case menu**
- **Type in a new name for the load case if required**
- **If the current load case is a factored case then type in the new load factors if desired**
- **Click on the OK button**

The name you give to the load case must not contain any of the following characters ; ^ ! < / (.

If you use the Edit Case command to change the name of a load case after analysis, your results will not be lost if you only change the name and not any load factors.

Editing Load Cases Numerically

You can edit the names and factors of load cases numerically in the Load Case table that may be displayed in the Data window. This table displays load case names and load case factors.

To change a load case name or factor

- **Click on the value to be changed**
- **Type in the new value**

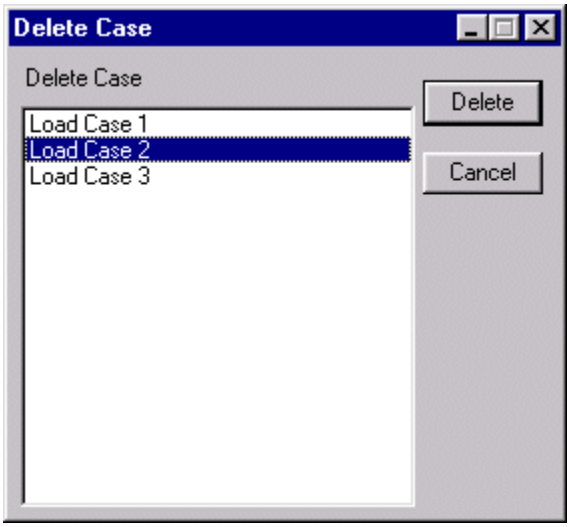
- **Press the enter key**

Deleting a Load Case

You can remove a load case from the structure. This will remove the case and all its loads. To delete a load case

- **Choose Delete Case from the Case menu**

A dialog box will appear with the current load cases in a list



- **Select the load case you wish to remove**
- **Click the Delete button**

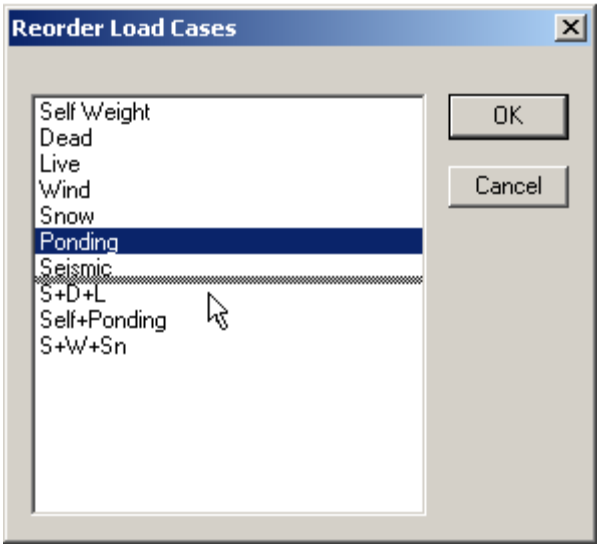
You select a load case in the list by clicking on its name. If you wish to remove more than one load case, hold down the shift key while clicking on the names. Holding down the shift key while clicking in the list adds the name to the selection or removes it if it is already selected.

Sorting Load Cases

You can change the order that the load cases appear in the list of cases. To sort the load cases

- **Choose Reorder Cases from the Case menu**

A dialog box will appear with the current load cases in a list.



Click and drag on a load case name to drag it to a different location in the list. Load cases which are used in a factored load case must appear before the factored case in the list.

Dynamic Loads

Multiframe allows you to apply dynamic loads to the structure which are used to perform a time history analysis.

Load Library

Multiframe4D includes a Load Library which contains commonly used dynamic loads. These loads may be spectra from earthquake measurements (several common earthquakes are provided), acceleration spectra from measured vibrations, and may also be dynamically varying forces which can be applied at joints in the structure.

When Multiframe4D starts up it will look for a file called "Load Library" or "Load Library.llb" in the same way that Multiframe looks for "Sections Library", first looking in the same directory as Multiframe4D, then looking on the current volume. Then if the Load Library is not found, Multiframe4D will prompt you for the location of the current load library.

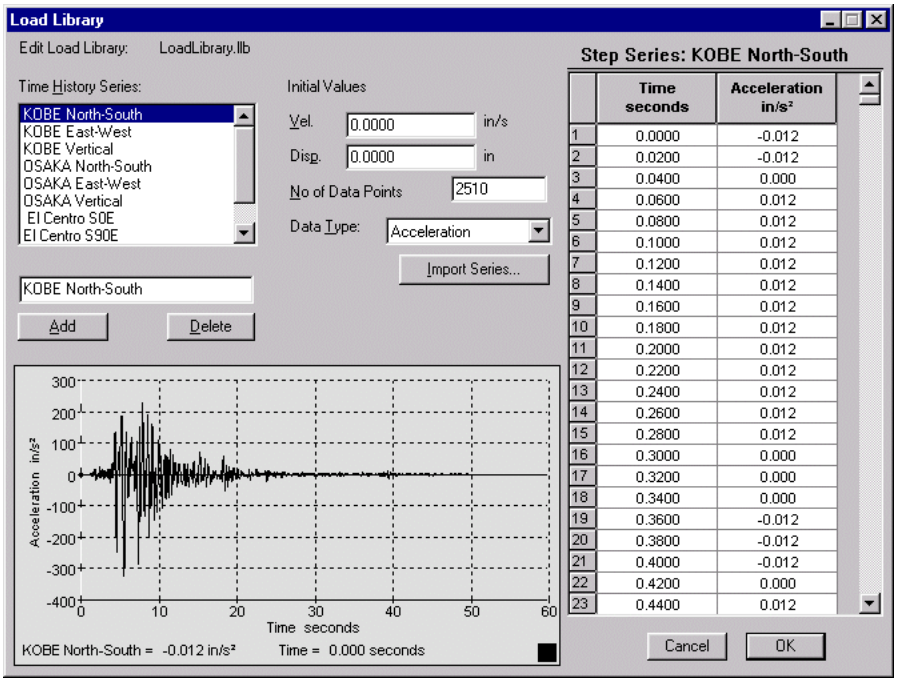
The Load Library shipping with Multiframe4D contains 9 earthquake spectra taken from 3 well known earthquakes. This includes accelerations in 3 orthogonal directions for each earthquake. The earthquakes are El Centro, Kobe Osaka.and Izmir, Turkey

Editing the Load Library

All dynamic loads are stored in the Load Library as either a force series or acceleration series. You can add, edit or delete loads in the Load Library.

- **Choose Edit Load Library... from the Edit menu**

The Load Library dialog will appear which displays the load and acceleration series currently stored in the library.



The list at the top left of the dialog lists all of the series stored in the Load Library. The name of the currently selected series is displayed below the list and may be edited there.

Each series has a number of attributes including start velocity and displacement, number of points and type of series (either force or acceleration).

To add a new series to the library

- **Click the Add button**
- **Type in a new name for the series**

Enter the attributes of the series and type the values for the series data in the table at the right hand side of the dialog. You can also copy and paste data into the table by selecting the cells to be pasted and using Paste in the Edit menu.

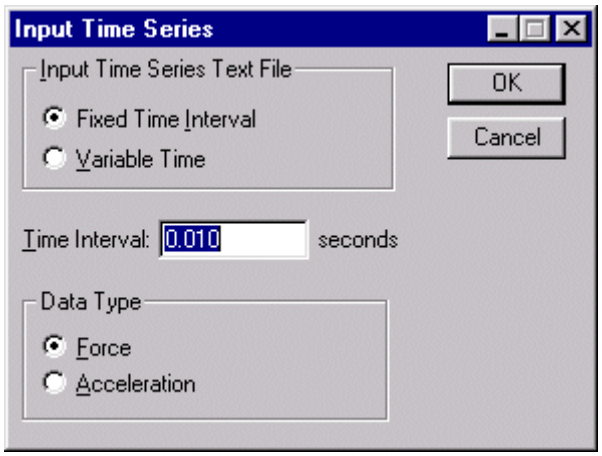
Remember that the final time value must be greater than any analysis time. Otherwise, the loading would be undefined. For a set load, the time could be any large number, as the load value does not vary.

Importing Load Data

You can import load data from a text file to the load library in addition to entering it by hand. To import load data

- **Click on the Import Series button**

A dialog will appear allowing you to choose the format of the data in the text file.



If you select Variable Time, the data should be in tab delimited format with two values per data point, the first value being the time in seconds and the second value being the load value (force or acceleration as appropriate).

If you select Fixed Time Interval, the data should be in tab delimited format with one force or acceleration value per data point.

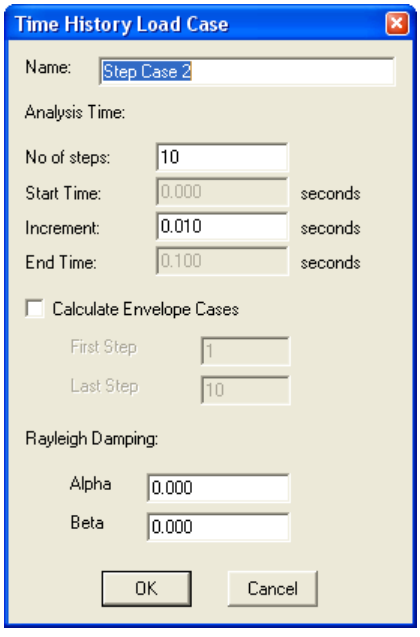
Adding Time History Load Cases

There are a number of different load cases that can be added in Multiframe. These include a self weight static load case, a normal static load case, a factored combination of static load cases, a Time History load case which can contain dynamically varying forces at one or more joints, and a seismic case which applies up to three orthogonal ground accelerations to any restrained joints.

To create a Time History Load Case

- **Select Time History... from the Add Load Case sub-menu**

A dialog will appear allowing you to enter the name, number and duration of the time steps for this load case along with other relevant information.

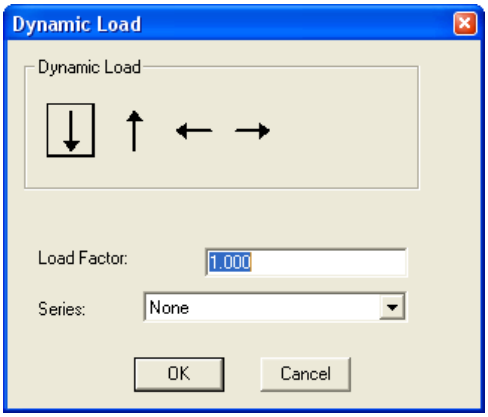


Applying Dynamic Loads

To add a dynamic load to the current Time History load case

- **Go to the Load window**
- **Select the joints to which you want to add the load**
- **Choose Dynamic Load from the Load menu**

This command is only available when a Time History loadcase has been created and selected. A dialog will appear allowing you to specify the direction of the load, a factor which will be multiplied by the values in the load series, and a pop-up to select the load series from the library.



- **Enter a load factor, if any**
- **Select the series you want to add and the global direction in which it is to act**

A force series needs to be created in the load library first. See Editing the Load Library for information on how to create a force series. Note that only force series may be applied when in a Time History load case and acceleration series can only be used in Seismic load cases.

Viewing Applied Time History Loads

To view the loads applied above while still in the current Time History load case

- **Choose Step Loads from the Data sub-menu under Display**

Data

	Node	Direction	Factor	Time History Series	Series Type
1	96	x	-1.000	Step Load	Force
2	25	z	-1.000	Step Load	Force
3	21	y	1.000	Step Load	Force

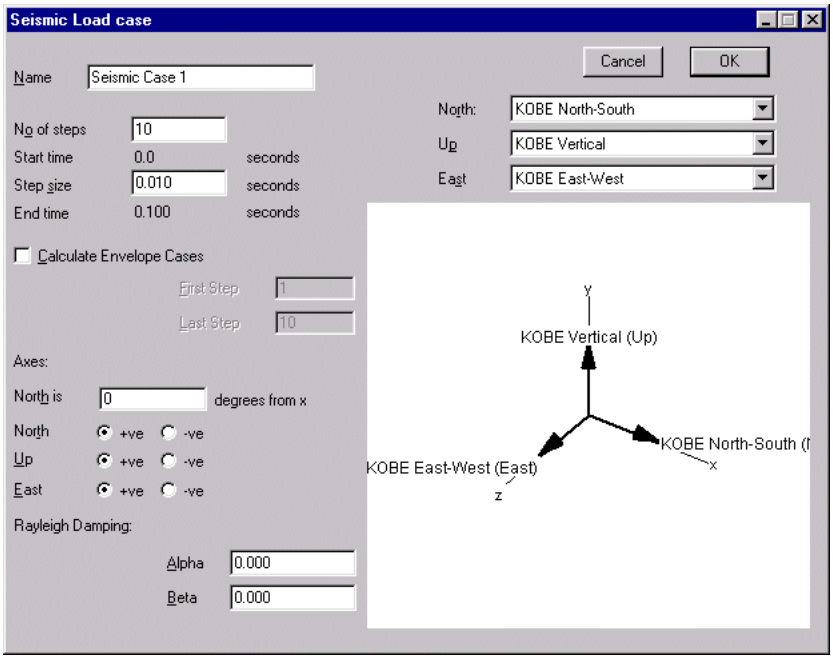
Adding Seismic Load Cases

In addition to creating Time History load cases that allow you to apply dynamically-varying forces to the structure, Multiframe4D allows you to create load cases that apply dynamically changing ground accelerations. The most common use for this is applying an earthquake spectrum to a frame, but it may also be used for any other ground-based acceleration. Analysis is carried out by applying the specified acceleration to all of the restrained joints in the structure. The assumption is that all restrained joints move with the ground acceleration. The results of the analysis are displayed relative to the ground position at each point in time. This means that the restrained joints all show zero deflection and the deflections of the frame are relative to the global axes as usual.

To create a Seismic Load Case

- **Select Seismic from the Add Load Case sub-menu under the Case menu**

A dialog will appear allowing you to enter the name and number and duration of time steps, acceleration series to be applied, and the direction of the acceleration.



- **Enter the name of the load case**
- **Enter the number of time steps to perform**
- **Enter the duration of each time step**

Multiframe will display the total duration of the analysis below these values.

- **Select the orientation of the axes along which the ground acceleration will be applied**

The normal convention for these axes is to label them **North**, **East** and **Up** for the three directions. You can specify the orientation of the North axis relative to the global x axis by entering an angle (counter-clockwise about the y axis) in degrees from the x axis. You can also reverse the direction of any of the load axes by choosing the appropriate radio button beside the axis name. The drawing in the bottom right hand corner of the dialog will display your selection.

Finally, you can choose which acceleration series from the Load Library will be applied to which axes by choosing the appropriate names from the pop-up menus.

Choosing the Time Step

The time step you choose for a time history analysis (**Δt**) has the effect of including or excluding the effects of natural frequencies modes in response of the structure. As **Δt** increases more modes are effectively excluded from the response behaviour as the time step. As a general rule if **T** is the period of the longest mode that is to be included in the response calculation you should choose **Δt** such that

$$\frac{\Delta t}{T} < 0.1$$

Rayleigh Damping Factors

When adding either a dynamic or seismic case the user can specify if proportional damping is to be used in the analysis. To do so you need to specify non-zero values for the Rayleigh damping coefficients Alpha and Beta. If you are uncertain as to how to find the appropriate value for Alpha and Beta please refer to texts on the subject. Finite Element Procedures in Engineering Analysis Bathe, Klaus-Jürgen © 1982 by Prentice Hall Inc. (pages 528-531) is a good reference.

Proportional damping assumes that the damping matrix is proportional to the mass and stiffness matrices M and K respectively.

$$C = \alpha \mathrm{M} + \beta \mathrm{K}$$

To calculate these values you need to look at the first two modes and the corresponding critical damping values required at those modes. Then using the equations

$$\alpha + \beta\omega_1^2 = 2\omega_1\xi_1$$

$$\alpha + \beta\omega_2^2 = 2\omega_2\xi_2$$

where ω_1 and ω_2 are the approximate first and second natural frequencies of the frame and ξ_1 and ξ_2 are the critical damping values corresponding to these frequencies .

Combing the equations gives

$$\alpha = \frac{2(\omega_2^2\omega_1\xi_1 - \omega_1^2\omega_2\xi_2)}{\omega_2^2 - \omega_1^2}$$

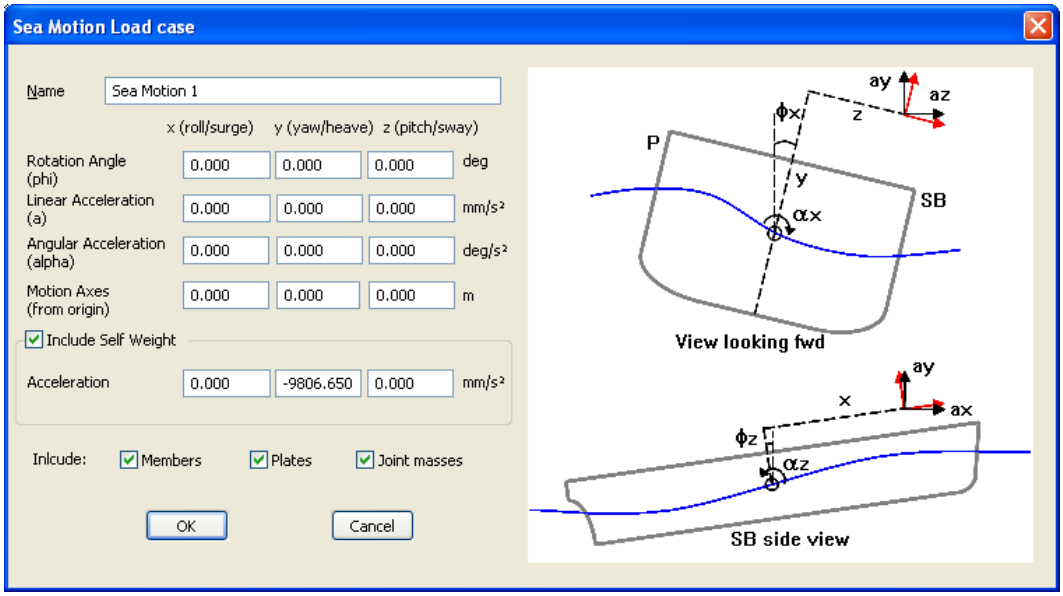
$$\beta = \frac{2(\omega_2\xi_2 - \omega_1\xi_1)}{\omega_2^2 - \omega_1^2}$$

Sea Motion Load Case

Multiframe allows you to add a Sea Motion load case, where linear and angular acceleration can be applied to a structure.

- **Choose Sea Motion… from the Add Case menu in the Case menu**

A dialog will be displayed which allows you to enter the accelerations due to the sea motion to be applied to the structure. Multiframe converts these into loads.



Rotation Angle (ϕ)

It is possible to model the structure heeled in any direction, or combination of directions, by entering values for Rotation Angle. In reality the structure is not rotated but the subsequent accelerations are applied at that angle.

Linear Acceleration (a)

Combinations of surge, heave and sway can be applied be entering values for Linear Acceleration. Surge is applied along the x-axis with forwards positive, heave is applied along the y-axis with upwards positive and sway is applied along the z-axis with to starboard positive.

Angular Acceleration (α)

Angular accelerations can be applied. These act around the centre of motion of the vessel, the location of which is defined by the Motion Axes, see below.

Roll is applied about the x-axis with starboard side down positive, yaw is applied about the y-axis with bow to starboard positive and pitch is applied about the z-axis with bow up positive.

Motion Axes

The location of the motion axes with respect to the origin is defined by the Motion Axes. The x direction is positive if the model's origin is forward of the centre of motion, the y direction is positive if above the centre of motion and the z direction positive if it is to the starboard of the centre of motion.

Self Weight

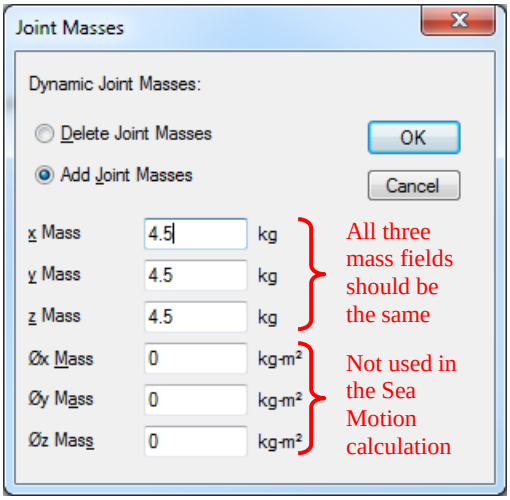
It is possible to include the Self Weight load case within the Sea Motion case. The reason for doing this, instead of creating a combined load case made up of a Sea Motion and a Self Weight load case, is because linear accelerations due to gravity will be adjusted for any Rotation Angles entered.

To include the self weight of the structure in the Sea Motion calculation tick the Include Self Weight check box and enter the gravitational acceleration. You will more than likely usually use the default setting of –g in the y direction.

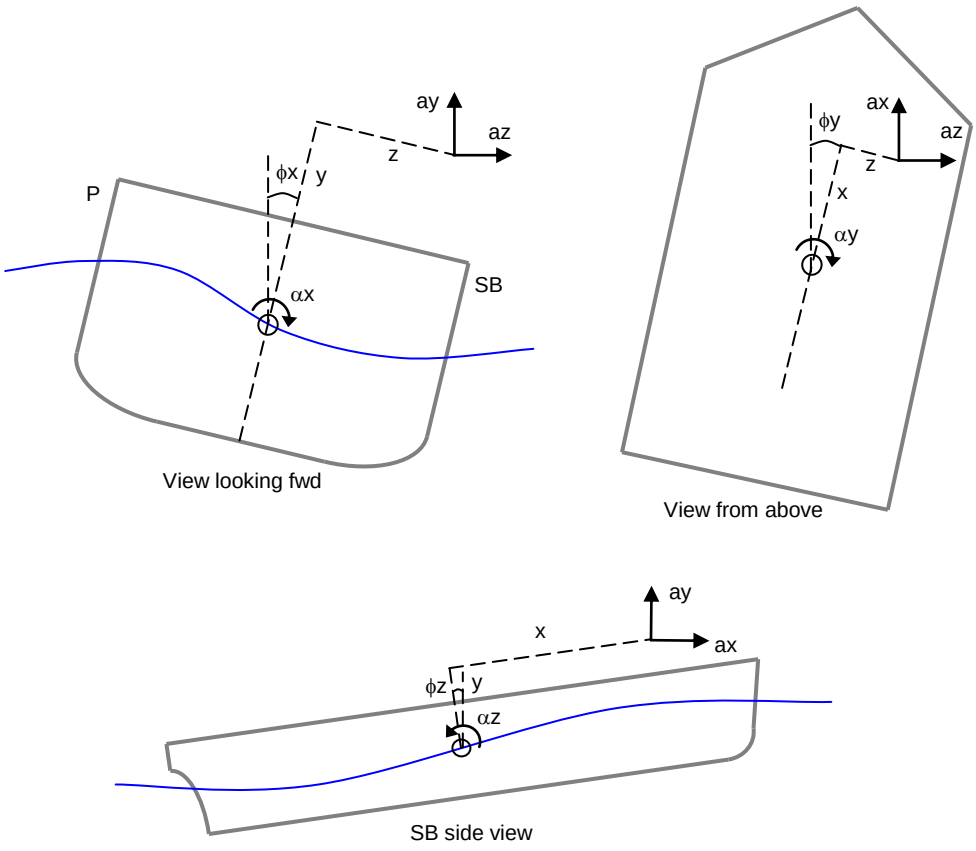
Include Members, Plates and Joint Masses

By un-ticking the checkboxes it is possible to exclude different types of element from the Sea Motion calculation.

Note that accelerations are only applied to Joint Masses in the direction that the Joint Mass acts. For the Joint Mass to act equally in all directions, enter the same value of mass in the x mass, y mass and z mass fields when adding the joint mass, as in the dialog below.



A summary of the Self Weight accelerations that can be applied and their positive directions can be seen in the following diagrams.



After clicking OK, a new Sea Motion load case will be created. You can create any number of sea motion load cases with different acceleration factors. You should make sure that you have set all of the section types for the structure before you use the Sea Motion command.

Sea Motion Loads

Sea Motion loads are applied and displayed similarly to Self Weight loads. To view the loads on the structure turn self weight loads on in the Display Symbols menu.

Sea Motion joint mass loads are applied as joint loads.

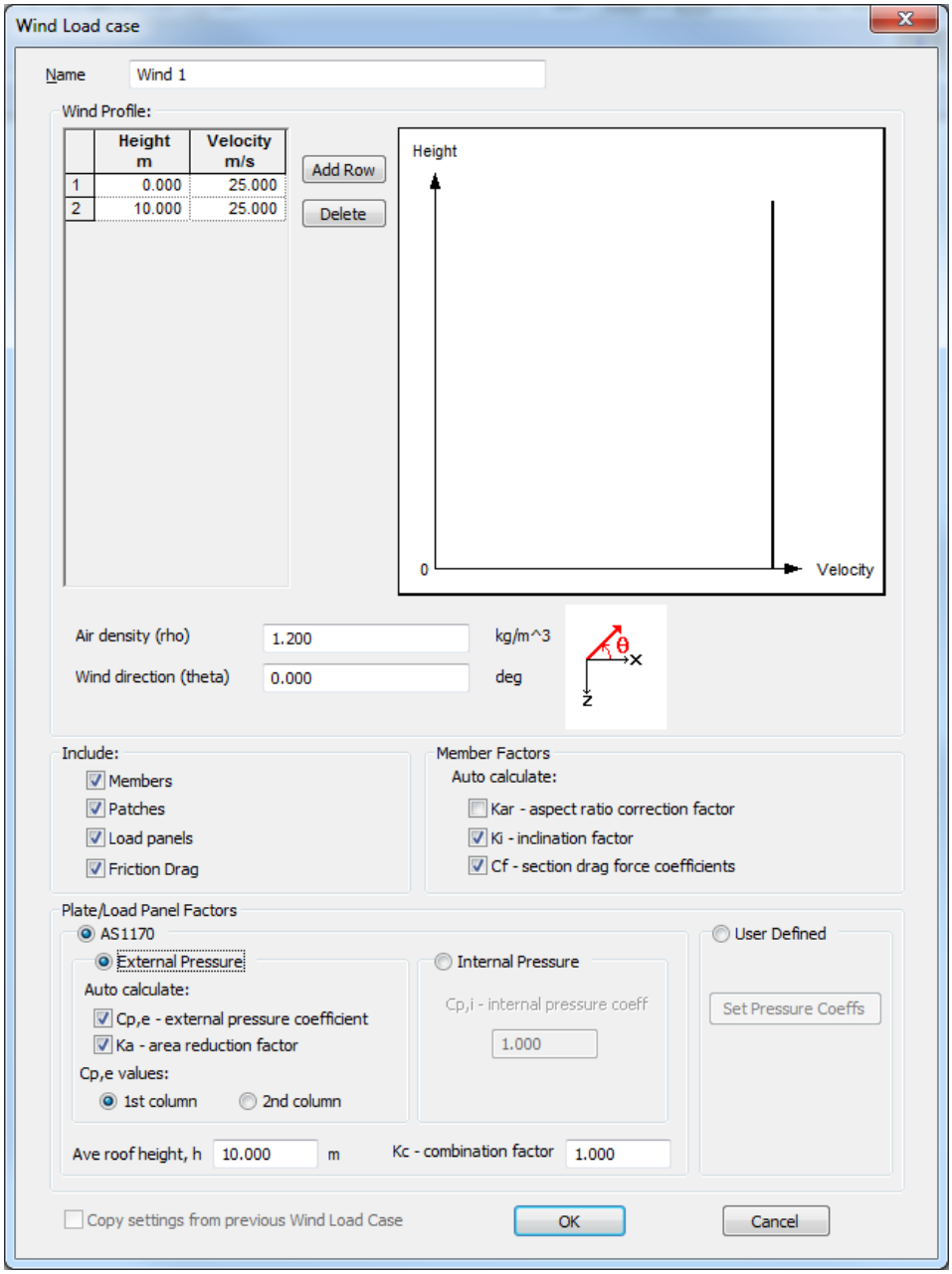
Wind Load Case

Multiframe 3D allows you to add a Wind load case, which automates the adding of wind loads from a defined to direction to the whole structure.

To create a Wind Load Case

- **Choose Wind… from the Add Case menu in the Case menu**

A dialog will be displayed which allows you to enter details of the wind strength and direction and which variables you would like to be automatically calculated. Multiframe converts this information loads on the structure.



Wind Profile

Define the wind velocity profile as pairs of height and velocity values. A height of zero is equivalent to the global axis origin ie y = 0. Use the Add Row button to increase the number of points that defines the profile. Select a row and press Delete to remove a row. The wind profile is shown graphically as you enter the values.

It is possible to enter negative heights if the origin is not at the base of your structure. The wind profile chart in the dialog will be updated accordingly and wind pressures applied to members with a negative height.

Air Density

Enter a value of air density. The wind pressure is calculated by

$P = 0.5*\rho_{air}*V^2*C$

Where C is a combination of the user and shape factors

Wind Direction

Enter the wind direction. The wind is applied horizontally in the x-z plane. A value of zero applied the wind along the x-axis. Positive angles of wind direction represent a rotation anti-clockwise.

Include Members/Patches/Load Panels

Tick the include check boxes to include/ignore any type of element from having a wind load applied.

Include Friction Drag

Tick the Friction Drag button to include friction drag force to sidewalls and roofs that run parallel with the defined wind direction. If Patches or Load Panels are not included the Friction Drag force will not be applied to them.

The Friction Drag Force is calculated by

$P = 0.5*\rho_{air}*V^2*C_f*C_u$

Where C_u is the user coefficient (default 1.0).

The default value of C_f, Friction Drag Coefficient, is taken to be 0.01. To change the value of C_f use the Patch/Load Panel Wind Factors dialog, as described below.

Member Factors

Tick the check boxes to define which member factors are automatically calculated. If the boxes are not ticked the default value is applied.
The member factors are:

- K_{ar} – Aspect ratio factor. As defined by AS1170 Appendix E Table E1. Defaults to 1.0.
- K_i – Inclination factor. As defined by AS1170 Appendix E2.1. Defaults to 1.0.
- C_f – Drag force coefficients. As defined by AS1170 Appendix E3. Defaults to 1.5.

Plate/Load Panel Factors – AS1170

Wind pressure loads on plates and load panels can be calculated based on Australian Standard AS1170.2.2002. To use this method select the AS1170 radio button and the following options will be made available:

External Pressure

External pressure loads are applied with the following settings:

Tick the check boxes to define which plate and load panel factors are automatically calculated. If the boxes are not ticked the default value is applied.
The plate/laod panel factors are:
K_a – Area reduction factor. As defined by AS1170 Table 5.4. Defaults to 1.0.
C_p – External pressure coefficient. As defined by AS1170 Chapter 5.4. Defaults to 0.8.
h – Average roof height. Define the average roof height of the structure at which the wind velocity for all patches/load panels will be taken.
$C_{p,e}$ values ($1^{st}/2^{nd}$ column) – choose between the External Pressure coefficients defined in Column 1 or 2 in Tables 5.3(A), 5.3(B) and Table D8.

Please note, currently Multiframe only supports stepped loads on patches but not load panels. Therefore when calculating the External Pressure Coefficients for Side Walls and Roofs with a slope less than 10 degrees on load panels, a weighted average pressure coefficient is calculated for the whole panel/patch.

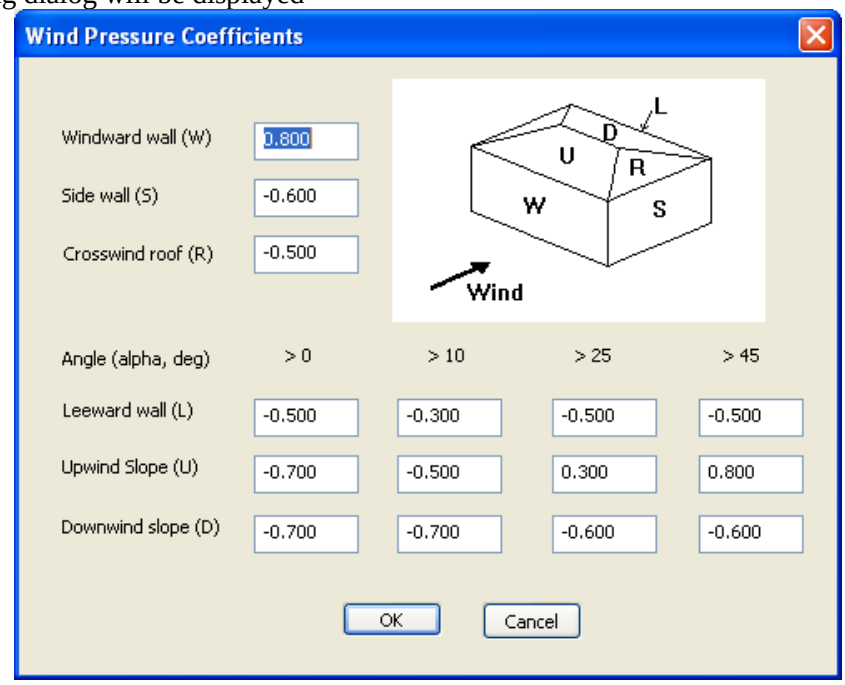
Internal Pressure

Internal pressure loads (normal to the patch/panels outwards from the centre of the model) are applied with the user defined pressure coefficient, $C_{p,i}$.

Plate/Load Panel Factors – User Defined

If you do not want to use the pressure coefficients as defined by AS1170, select the radio button labelled User Defined.

To define your own pressure coefficients, press the button Set Pressure Coeffs. The following dialog will be displayed



Enter values of pressure coefficient for each of the sides of your structure as described in the diagram. For sloping sides it is possible to define different pressure coefficients for varying angles of slope.

Copy settings from previous Wind Load Case

If you have already defined a wind load case it is possible to copy the settings from that load case by ticking the check box at the bottom of the add Wind load case dialog.

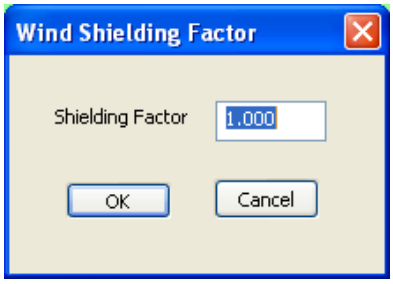
Member Wind Shielding Factor

Edit the member wind shielding factor to account for any shielding within the structure. One or more shielding factors can be changed at the same time by selecting multiple members.

To edit the member wind shielding factor

- **Select one or more members**
- **Choose Member Wind Shielding Factor… from the Case menu**

The following dialog will be displayed



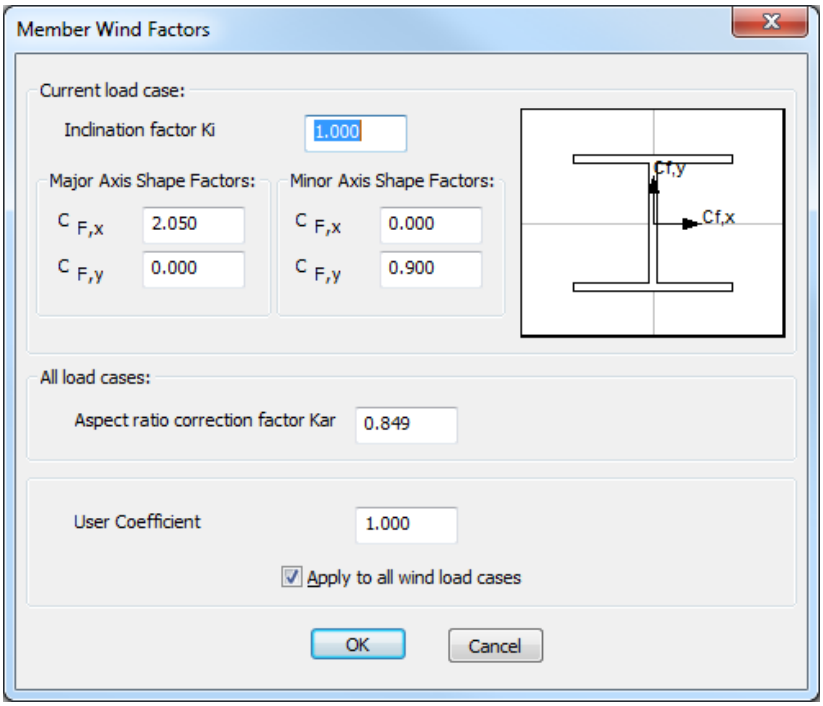
Member Wind Factors

The automatically calculated member wind factors for each individual member can be edited via the Member Wind Factors dialog.

To edit the member wind factors

- **Select one or more members**
- **Choose Member Wind Factors… from the Case menu**

The following dialog will be displayed



As well as allowing the factors to be changed the orientation of the section to the wind is displayed.

The Wind force is resolved into local member loads. Thus major and minor axis loads are applied in the directions shown in the dialog. Each of these loads has a pair of shape factors associated with them.

Please note that for symmetrical sections, the transverse shape factor for each direction ie $C_{F,y}$ for Major axis loading and $C_{F,x}$ for Minor axis loading, is set to zero instead of applying two transverse loads in opposing directions, which would cancel each other out.

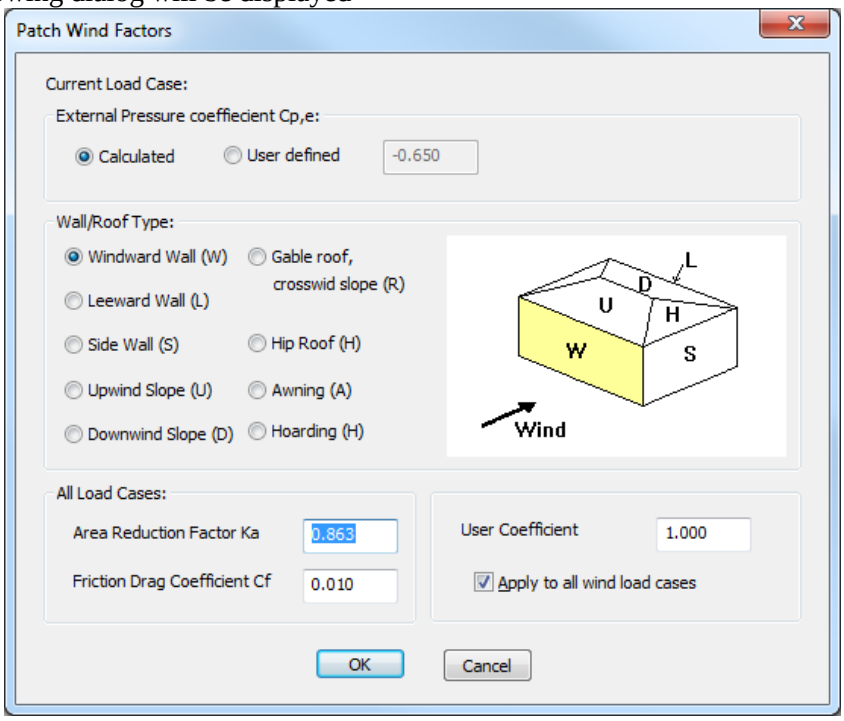
Patch Wind Factors

The automatically calculated patch wind factors for each individual patch can be edited via the Patch Wind Factors dialog.

To edit the patch wind factors

- **Select one or more patches**
- **Choose Patch Wind Factors… from the Case menu**

The following dialog will be displayed



The wall/roof type for the selected patch is indicated by the radio buttons and diagram. If Multiframe has incorrectly identified a wall it is possible to overwrite the selection by clicking one of the other wall types. If the External Pressure coefficient selection is set to Calculated the value of $C_{p,e}$ will be automatically updated.

Please note, Multiframe is unable to automatically identify Awnings or Hoardings. These must be manually identified using this dialog.

Load Panel Wind Factors

The load panel wind factors can be edited via the load panel wind factors dialog.

To edit the load panel wind factors

- **Select one or more load panels**
- **Choose Load Panel Wind Factors… from the Case menu**

A similar dialog to the Patch Wind Factors will be displayed.

Display Load Panel/Patch Wind Pressure Coefficients

It is possible to display the Load Panel and Patch wind pressure coefficients directly on to the panel/patch that they refer to. To do this.

- **Choose Symbols… from the Display menu**

- **Select Pressure Coeff from either the Panel or Patch group**

To view the value clearly you may have to turn off the panel/patch loads and any other labels.

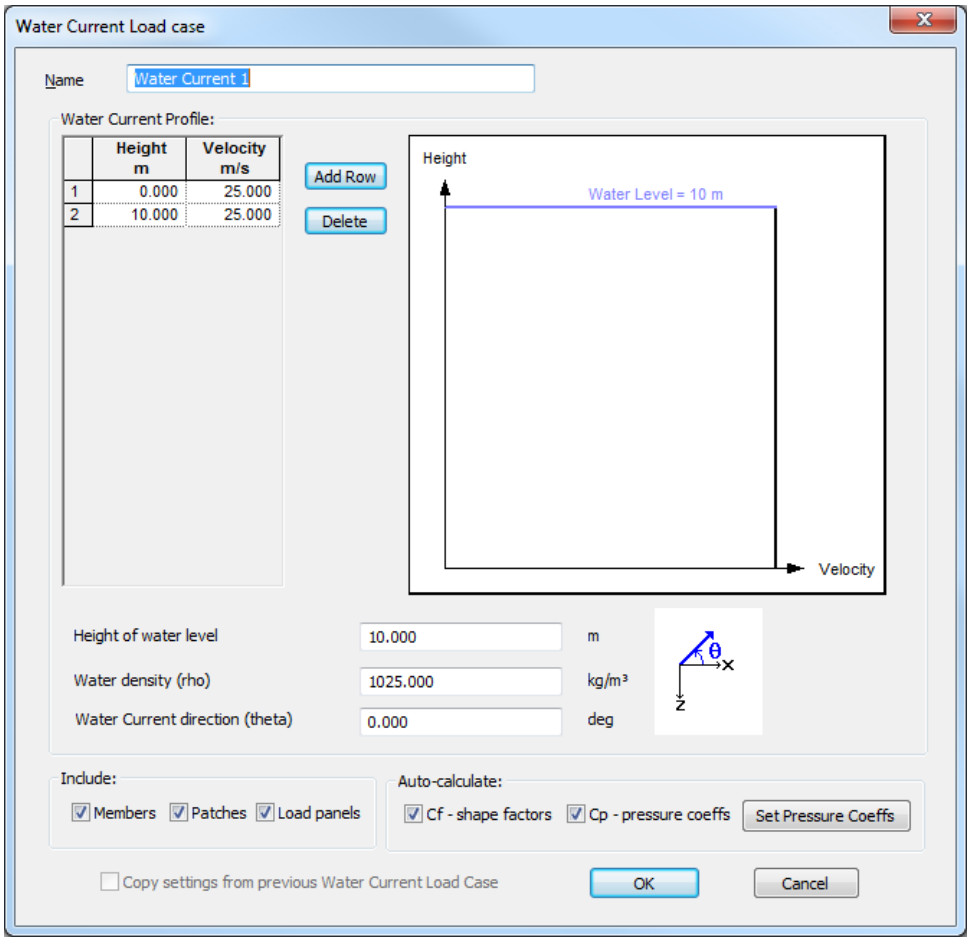
Water Current Load Case

Multiframe 3D allows you to add a Water Current load case, which automates the adding of fluid loads from a defined to direction to the whole structure.

To create a Water Current Load Case

- **Choose Water Current… from the Add Case menu in the Case menu**

A dialog will be displayed which allows you to enter details of the current strength and direction and which variables you would like to be automatically calculated. Multiframe converts this information loads on the structure.



Water Current Profile

Define the water current velocity profile as pairs of height and velocity values. A height of zero is equivalent to the global axis origin ie y = 0. Use the Add Row button to increase the number of points that defines the profile. Select a row and press Delete to remove a row. The current profile is shown graphically as you enter the values.

Height of Water Level

Enter the depth of the water above the origin. Any part of the structure that protrudes above the surface of the water will not have a Water Current load applied to it.

Water Density

Enter a value of water density. The water current pressure is calculated by

$P = 0.5*\rho_{water}*V^2*C$

Where C is a combination of the user and shape factors

Water Current Direction

Enter the water current direction. The water current is applied horizontally in the x-z plane. A value of zero applied the water current along the x-axis. Positive angles of water current direction represent a rotation anti-clockwise.

Include Members/Patches/Load Panels

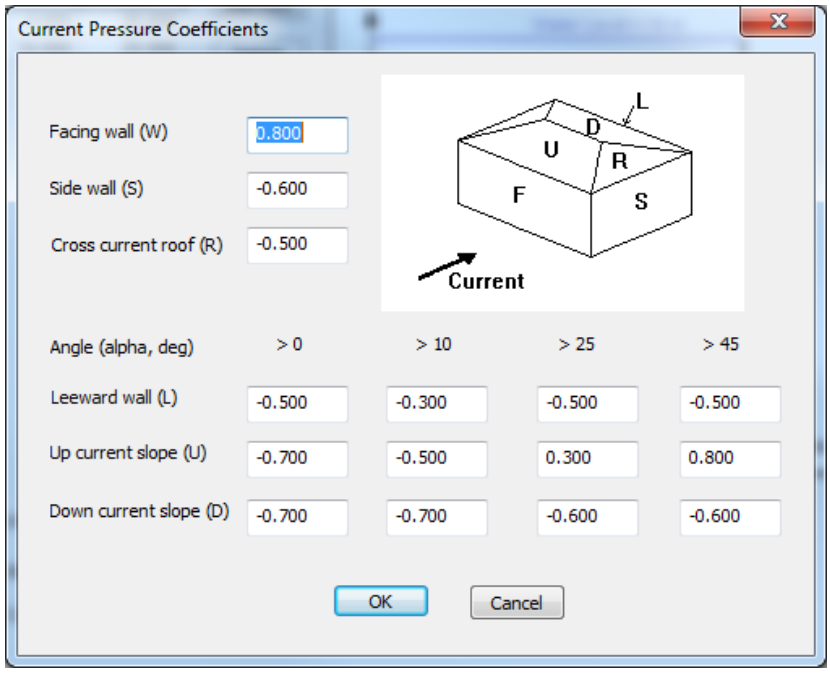
Tick the include check boxes to include/ignore any type of element from having a water current load applied.

Auto-calculate

Tick the check boxes to define which factors are automatically calculated. If the boxes are not ticked the default value is applied.
The member factors are:

- C_f – Drag force coefficients. As defined by AS1170 Appendix E3. Defaults to 1.5.
- C_p – External pressure coefficient as defined with the Set Pressure Coeffs button. Defailts to 0.8.

To define your own pressure coefficients, press the button Set Pressure Coeffs. The following dialog will be displayed



Enter values of pressure coefficient for each of the sides of your structure as described in the diagram. For sloping sides it is possible to define different pressure coefficients for varying angles of slope.

Copy settings from previous Water Current Load Case

If you have already defined a water current load case it is possible to copy the settings from that load case by ticking the check box at the bottom of the add Water Current load case dialog.

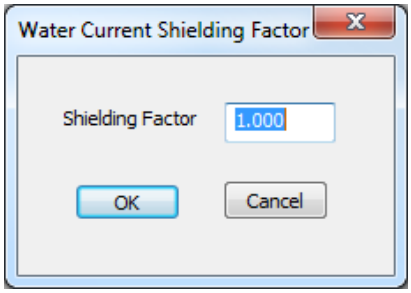
Member Water Current Shielding Factor

Edit the member water current shielding factor to account for any shielding within the structure. One or more shielding factors can be changed at the same time by selecting multiple members.

To edit the member water current shielding factor

- **Select one or more members**
- **Choose Member Water Current Shielding Factor… from the Case menu**

The following dialog will be displayed



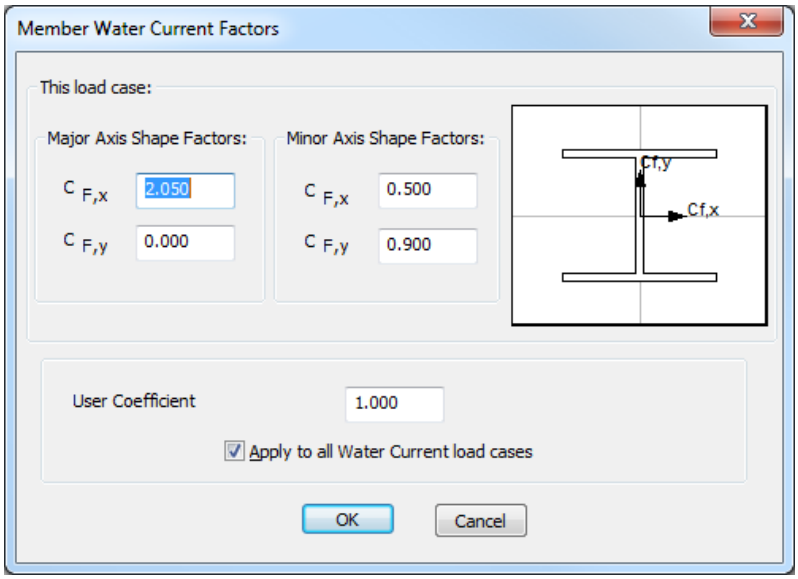
Member Water Current Factors

The automatically calculated member water current factors for each individual member can be edited via the Member Water Current Factors dialog.

To edit the member water current factors

- **Select one or more members**
- **Choose Member Water Current Factors… from the Case menu**

The following dialog will be displayed



As well as allowing the factors to be changed the orientation of the section is displayed.

Water Currents are resolved into local member loads. Thus major and minor axis loads are applied in the directions shown in the dialog. Each of these loads has a pair of shape factors associated with them.

Please note that for symmetrical sections, the transverse shape factor for each direction ie $C_{F,y}$ for Major axis loading and $C_{F,x}$ for Minor axis loading, is set to zero instead of applying two transverse loads in opposing directions, which would cancel each other out.

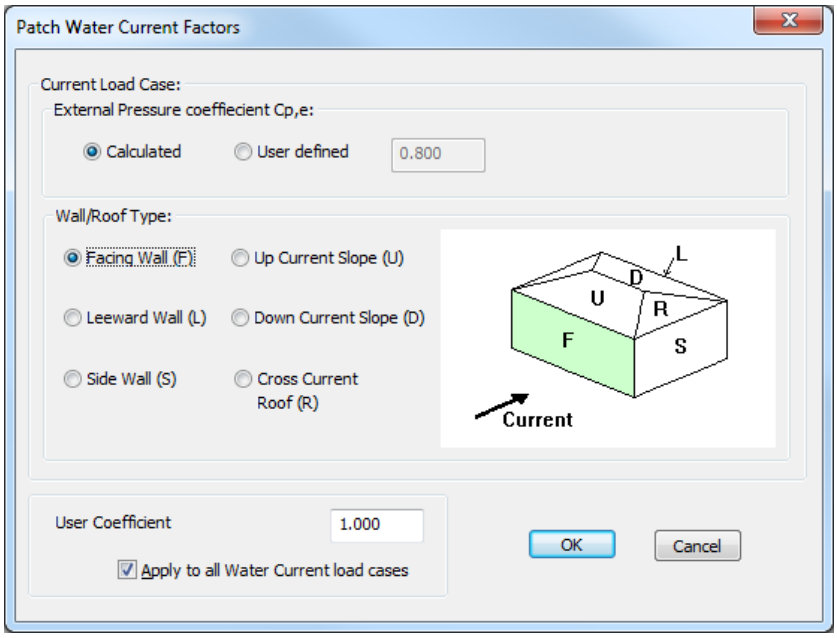
Patch Water Current Factors

The automatically calculated patch water current factors for each individual patch can be edited via the Patch Water Current Factors dialog.

To edit the patch water current factors

- **Select one or more patches**
- **Choose Patch Water Current Factors… from the Case menu**

The following dialog will be displayed



The wall/roof type for the selected patch is indicated by the radio buttons and diagram. If Multiframe has incorrectly identified a wall it is possible to overwrite the selection by clicking one of the other wall types. If the External Pressure coefficient selection is set to Calculated the value of $C_{p,e}$ will be automatically updated.

Load Panel Water Current Factors

The load panel water current factors can be edited via the load panel water current factors dialog.

To edit the load panel water current factors

- **Select one or more load panels**
- **Choose Load Panel Water Current Factors… from the Case menu**

A similar dialog to the Patch Water Current Factors will be displayed.

Display Load Panel/Patch Water Current Pressure Coefficients

It is possible to display the Load Panel and Patch water current pressure coefficients directly on to the panel/patch that they refer to. To do this.

- **Choose Symbols… from the Display menu**
- **Select Pressure Coeff from either the Panel or Patch group**

To view the value clearly you may have to turn off the panel/patch loads and any other labels.

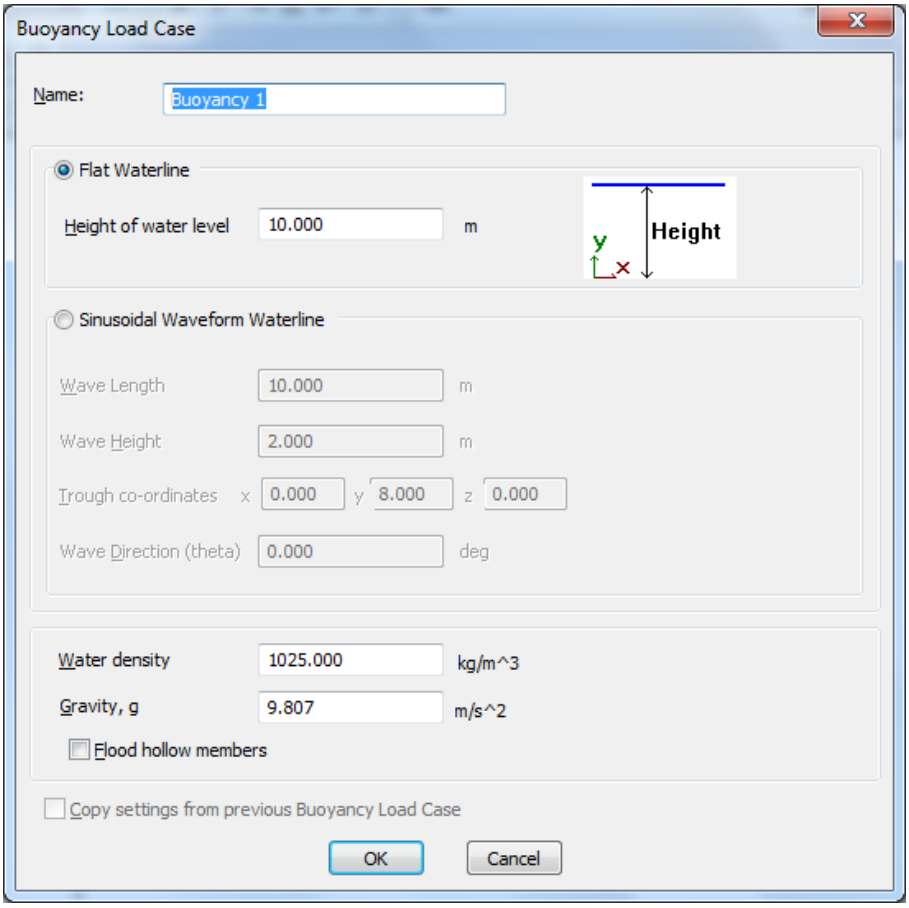
Buoyancy Load Case

Multiframe allows you to add a Buoyancy load case, which automates the applying of fluid buoyancy loads to the whole structure.

To create a Buoyancy Load Case

- **Choose Buoyancy… from the Add Case menu in the Case menu**

A dialog will be displayed which allows you to enter details of the buoyancy load case.



Flat/Sinusoidal Waterline

Select either a flat or a sinusoidal wave fluid surface.

Height of Water Level

For a flat waterline, enter the height of the water level above the origin. Any part of the structure that protrudes above the surface of the water will not have a Buoyancy load applied to it.

Wave Length

For a sinusoidal waterline, enter a value for the wave length.

Wave Height

For a sinusoidal waterline, enter a value for the wave height.

Trough Co-ordinates

For a sinusoidal waterline, enter values for the wave trough co-ordinates.

Wave Direction

For a sinusoidal waterline, enter the wave direction. The wave is applied horizontally in the x-z plane. A value of zero applied the water current along the x-axis. Positive angles of water current direction represent a rotation anti-clockwise.

Water Density

Enter a value of water density.

Gravity

Enter the value of gravitational acceleration.

Flood Hollow Members

Tick the check box to flood any hollow (SHS, RHS, tubes) members. If this button is un-checked the buoyancy of such members is taken to be that of their total enclosed volume.

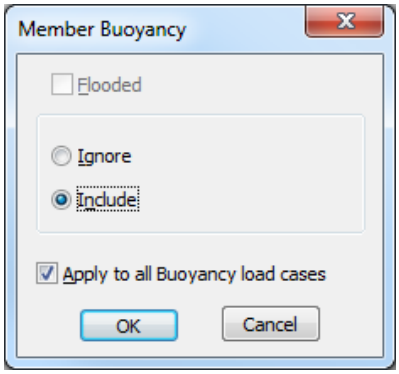
Member Buoyancy

Individual member buoyancy factors can be edited by using the Member Buoyancy dialog.

To edit the member buoyancy

- **Select one or more members**
- **Choose Member Buoyancy… from the Case menu**

The following dialog will be displayed



Members can be completely ignored (no buoyancy load applied) or their flooded status set.

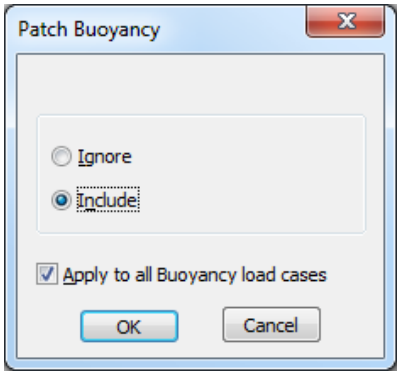
Patch Buoyancy

Individual patch buoyancy factors can be edited by using the Patch Buoyancy dialog.

To edit the patch buoyancy

- **Select one or more patches**
- **Choose Patch Buoyancy… from the Case menu**

The following dialog will be displayed



Patches can be completely ignored (no buoyancy load applied).

Plate Buoyancy

A dialog similar to that for patch buoyancy can be used to edit individual plate buoyancy factors. Plate selection must be turned on and one or more plate selected to access this dialog.

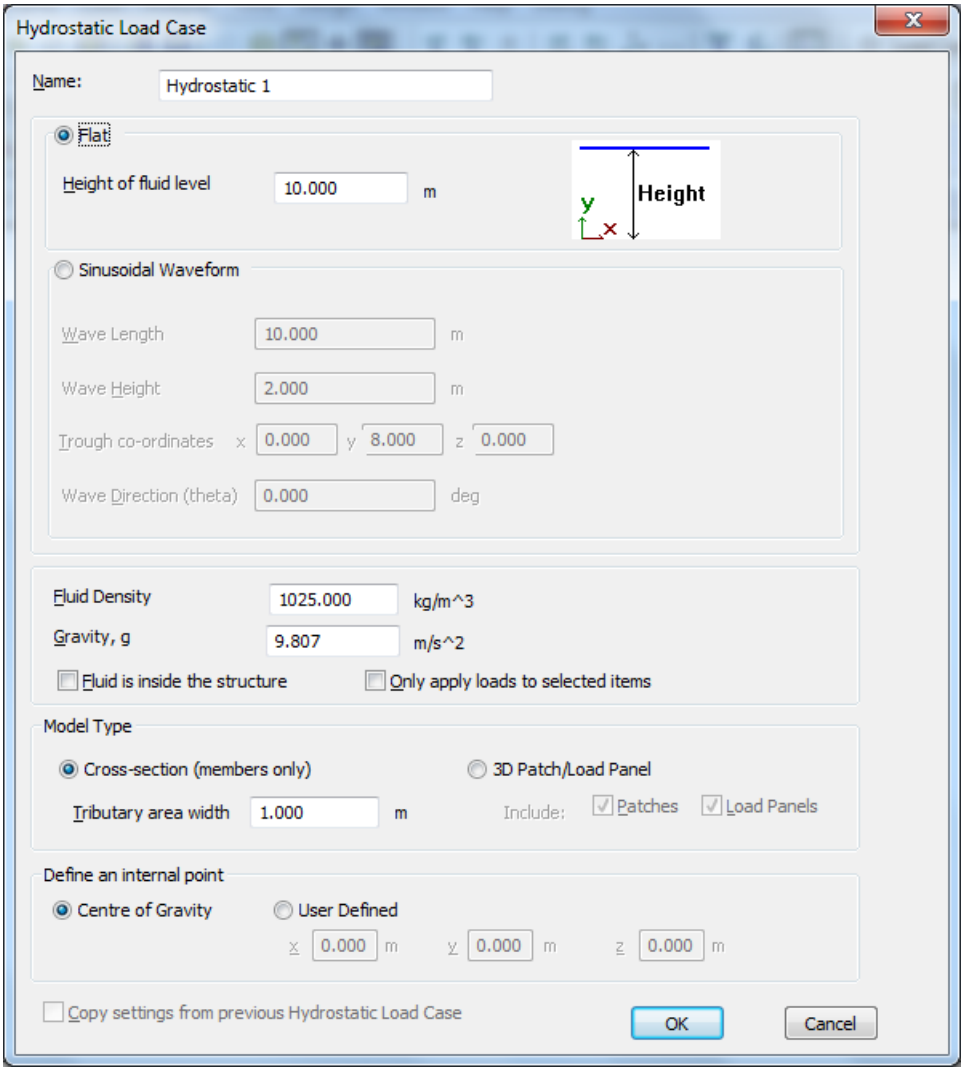
Hydrostatic Load Case

Multiframe allows you to add a Hydrostatic load case, which automates the applying of fluid hydrostatic loads to the structure.

To create a Hydrostatic Load Case

- **Choose Hydrostatic… from the Add Case menu in the Case menu**

A dialog will be displayed which allows you to enter details of the hydrostatic load case.



Flat/Sinusoidal Waveform

Select either a flat or a sinusoidal wave fluid surface.

Height of Fluid Level

For a flat fluid surface, enter the height of the fluid level above the origin. Any part of the structure that protrudes above the surface of the fluid will not have a Hydrostatic load applied to it.

Wave Length

For a sinusoidal wave fluid surface, enter a value for the wave length.

Wave Height

For a sinusoidal wave fluid surface, enter a value for the wave height.

Trough Co-ordinates

For a sinusoidal wave fluid surface, enter values for the wave trough co-ordinates.

Wave Direction

For a sinusoidal wave fluid surface, enter the wave direction. The wave is applied horizontally in the x-z plane. A value of zero applied the water current along the x-axis. Positive angles of water current direction represent a rotation anti-clockwise.

Fluid Density

Enter a value of fluid density.

Gravity

Enter the value of gravitational acceleration.

Fluid is Inside the Structure

Tick this check box if you are modelling a tank or similar that contains a fluid ie the pressure is pushing out. Leave it un-ticked if you and modelling a structure that is submerged in a fluid ie a boat.

Only apply loads to selected items

Tick this check box if you only want to apply hydrostatic loads to the patches, panels and elements that you had selected in the Load window before opening the Add Hydrostatic Load Case dialog.
If you leave this check box un-ticked, Hydrostatic loads will be applied to the whole structure.

Model Type

The hydrostatic load case is intended for two different types of model:

1. A ship hull cross-section made up of members only. In this case the tributary area width is required that the fluid pressure is applied over. This would typically be the length of one frame spacing.
2. A 3D patch/load panel model where the fluid pressure is applied to the patches/panels ie a whole ship or a tank.

Multiframe will detect if there are any patches or load panels in the model and choose the appropriate model type accordingly. If this selection is wrong the user can override it.

Define an Internal Point

In order that the loads are applied in the correct direction, a internal point must be defined inside the structure. All the loads will be applied to/from thus point, depending whether the fluid is inside or outside of the structure.

As a first guess, Multiframe uses the centre of gravity of the structure. If this not correct, the user can define a point themselves.

Copy settings from previous Hydrostatic Load Case

If you have already defined a hydrostatic load case it is possible to copy the settings from that load case by ticking the check box at the bottom of the add Hydrostatic load case dialog.

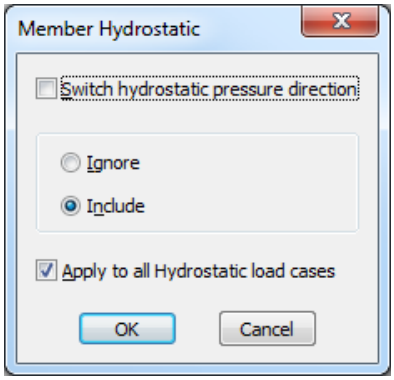
Member Hydrostatic

Individual member hydrostatic factors can be edited by using the Member Hydrostatic dialog.

To edit the member hydrostatic

- **Select one or more members**
- **Choose Member Hydrostatic… from the Case menu**

The following dialog will be displayed



Members can be completely ignored (no hydrostatic load applied) or the direction of the hydrostatic load switched.

Patch, Plate and Load Panel Hydrostatic

A dialog similar to that for member hydrostatic can be used to edit individual patch, plate or load panel hydrostatic factors. Plate selection must be turned on and one or more plate selected to access the plate hydrostatic dialog.

Evaluating Expressions

Many of the dialog fields in Multiframe allow you to enter a math expression instead of a real number. The mathematical expressions can contain the usual mathematical operators (+.-,*,/) as well as some common mathematical functions. In addition, expressions can also contain a number of predefined variables that describe the properties of joints or members, For example, the variable L is used to represent the length of a member in dialogs that act upon a selection of members.

Intrinsic Functions

The following functions are supported by Multiframe when evaluating mathematical expressions.

Function	Description	Units
Sin(*x*)	Sine of *x*	*x* in Radians
Cos(*x*)	Cosine of *x*	*x* in Radians
Tan(*x*)	Tangent of *x*	*x* in Radians
Arcsin(*x*)	Arcsine of *x*	Degrees
Arccos(*x*)	Arccosine of *x*	Degrees
Arctan(*x*)	Arctangent of *x*	Degrees
Asin(*x*)	Arcsine of *x*	Radians
Acos(*x*)	Arccosine of *x*	Radians
Atan(*x*)	Arctangent of *x*	Radians
Atan2(x,y)	Returns arctangent of x/y in the range from -180° to +180°	
Arctan2(x,y)	Returns arctangent of x/y in the range from $-\pi$ to $+\pi$	
Sqr(*x*)	Square of *x*	
Sqrt(*x*)	Square root of *x*	

Exp(*x*))	Exponential of *x*
Ln(*x*)	Natural logarithm of *x*
Log(*x*)	Base 10 logarithm of *x*
Abs(*x*)	Absolute value of *x*
Max(x,y)	Returns maximum value of x and y
Min(x,y)	Returns minimum value of x and y

Special Constants

The following constants are supported by Multiframe when evaluating mathematical expressions.

Function	Description	Units
Pi	Value of Pi (3.14159…)	

Joint Variables

In dialogs that act upon joint loads, restraints or the properties of a joint, Multiframe can evaluate expressions that include variables representing the coordinates of a joint. These variables are as follows:

Variable	Description	Units
X	x coordinate of joint	Length
Y	y coordinate of joint	Length
Z	z coordinate of joint	Length

When the dialog is accepted, each joint in the current selection is considered in turn, the above variables substituted by the location of the joint, and the appropriate values set for the load, restraint or joint property. The following dialogs currently support evaluation of these variables:

- **Joint Spring dialog**
- **Joint Displacement dialog**
- **Joint Orientation dialog**
- **Joint Load dialogs**
- **Joint Moment dialogs**
- **Dynamic Load dialog**

Member Variables

Multiframe can evaluate expressions that contain variables which represent the geometric properties of a member or it's section properties. These variables include:

Variable	Description	Units
L	Length of member	Length
X1	x coord of joint at start of member	Length
Y1	y coord of joint at start of member	Length
Z1	z coord of joint at start of member	Length
X2	x coord of joint at end of member	Length
Y2	y coord of joint at end of member	Length
Z2	z coord of joint at end of member	Length
B	Width of section	Displacement
D	Depth of section	Displacement

Tf	Flange thickness	Displacement
Tw	Web thickness	Displacement
A	Area of section	Area
theta	Member Orientation	radians

In addition, the section properties as described in the CalcSheet section of the manual can also be evaluated. When a dialog is accepted, each member in the current selection is considered in turn, the above variables are substituted by the appropriate value, and the expressions evaluated before setting the value of the field.

The following dialogs support parsing member variables:

- **Member Offsets dialog**
- **Member Orientation dialog**
- **All Member load dialogs**
- **Bending dialog (Steel Designer)**
- **Tension dialog (Steel Designer)**
- **Compression dialog (Steel Designer)**
- **Constraints dialog (Steel Designer)**
- **Serviceability dialog (Steel Designer, AS4100)**

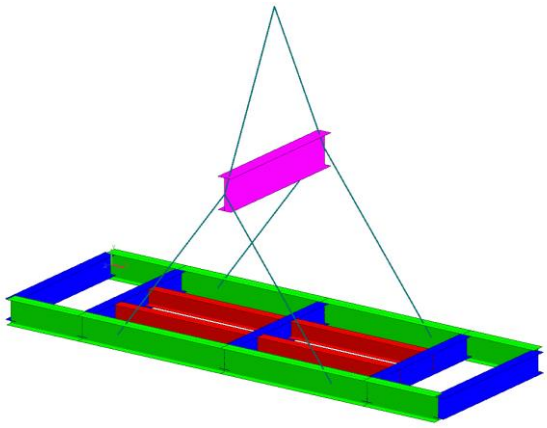
Analysis

When you have finished setting up your structure and its restraints and loading, you use the Analyse menu to carry out structural analysis on the structure. A progress bar will be displayed while the analysis is in progress.

Multiframe will check the structure prior to analysis and alert you if there are any problems. For example: You must have specified the section type for all of the members. If you have missed any members, Multiframe will alert you with a dialog and will select all of the undefined members in the Frame window. You will also be alerted if there are any mechanisms or there are unrestrained degrees of freedom. In each case the offending part of the structure will be selected in the Frame window.

Different analysis types are available dependent on the version you are using. This is summarised in the table below. Click on the links on the left for more information about the specific analysis type.

	Multiframe 2D	**Multiframe 3D**	**Multifra 4D**
Linear Analysis	✓	✓	✓
Nonlinear Analysis	✓	✓	✓
A new type of analysis has been added for analysing frames which are used for lifting calculations. Typical examples would be lifting of baskets and skids with slings which are attached to a crane.	✗	✓	✓



Such models are typified by a lack of restraint in the horizontal plane which allows the frame to either rotate or swing freely. If you model a frame like this you will often get the error message “Solution does not make sense”. The lifting analysis function automatically adds additional spring restraints to the model to provide sufficient horizontal restraint to allow the model to be analysed. This eliminates the need for you to apply your own manual spring restraints.

The stiffness of these restraints is automatically set by Multiframe so as to have a minimal effect on the actions within the frame. They are set at a small proportion of the other stiffnesses in the frame. While you are creating your structural model, you should review the location of the centre of mass and ensure that the lifting point is located as near as possible to a location directly above the centre of mass of the structure and its loads.

Note also that when modelling slings or lifting cables you should release both moments and torsion at the ends of those members. If you release all of the sling members connecting to the lifting point, be sure to applied a fixed restraint to the lifting point to prevent that node from rotating.

A lifting analysis is performed via the Lifting… command in the Analysis menu. A lifting analysis is in fact a special type of linear analysis and as such the results of the lifting analysis are stored as the results of a linear analysis. If your sling members are set to Tension-Only, then a dialog will appear allowing you to take those effects into account during analysis. You are no longer required to use a NonLinear analysis for Tension-Only effects.

Some care must be taken in performing a lifting analysis as the additional spring restraints added for the analysis will also help to suppress structural mechanisms. After analysis is complete, you will notice that all nodes in the model will have reactions. You should see normal reactions at any nodes where you have applied a restraint, and at other nodes where automatic springs have been applied, you should see reactions with a small value. These small reactions are a consequence of the automatically applied springs. If you see unacceptably large reactions at any of these nodes, then you should use manual spring restraints and carry out a normal linear analysis.

Buckling Analysis

Modal Analysis			
Time History Analysis			
Batch Analysis			

Also see:

Viewing Results on page 195.
Chapter 4 Multiframe Analysis on page 297 for information on the calculation methods used in Multiframe and sign conventions etcetera.

Precision and Memory Requirements

The solver will automatically switch from double precision to single precision arithmetic if there is not enough memory. Single precision arithmetic is less precise than double (it has about 7 digits of precision as opposed to 15 digits for double), but has the advantage of requiring less storage space for the stiffness matrix. This allows you to solve larger problems or have more load cases for the same amount of memory. The only problem that can arise from the use of single precision arithmetic, is that if the stiffness matrix is badly conditioned, round-off errors may accumulate during analysis and may make results less accurate. Multiframe will warn you if it switches to single precision and give you the option of analysing using single precision or cancelling the analysis.

The solver includes error checking routines to detect problems with the structure. If you have drawn any members that are not connected to the rest of the structure, Multiframe will alert you to this fact. The section properties of all the sections will also be checked to ensure that they are non-zero where appropriate.

Reporting

The Reporting group at the bottom of the Analysis dialog contains three buttons for selecting the level of reporting to be output to the Report Window during an analysis. Reporting is only used for a nonlinear analysis for which convergence and iteration data are output to the Report Window in addition to any warnings and error messages in relation to the analysis. The Full report also includes the displacement and force norms at the end of each iteration.

Cancelling Analyses

A progress indicator is displayed while analysis is in progress. To cancel analysis

- **Hit the escape key, or**
- **Press cancel in the progress dialog**

The cursor will change to a watch and analysis will be terminated.

You can switch from Multiframe to another program while analysis is in progress.

Linear Analysis

To perform a linear analysis

- **Choose Analyse Linear from the Analyse menu or press the F2 function key on your keyboard**

Multiframe will immediately display the progress dialog and proceed with the analysis of all load cases.

Nonlinear Analysis

Nonlinear Analysis – Concepts

Nonlinear analysis considers second order elastic nonlinearities that are due to the P-δ, P-Δ, and flexural shortening effects. The nonlinear analysis also accounts for the influence of any tension or compression only members within a structure.

Nonlinear Analysis – Procedures

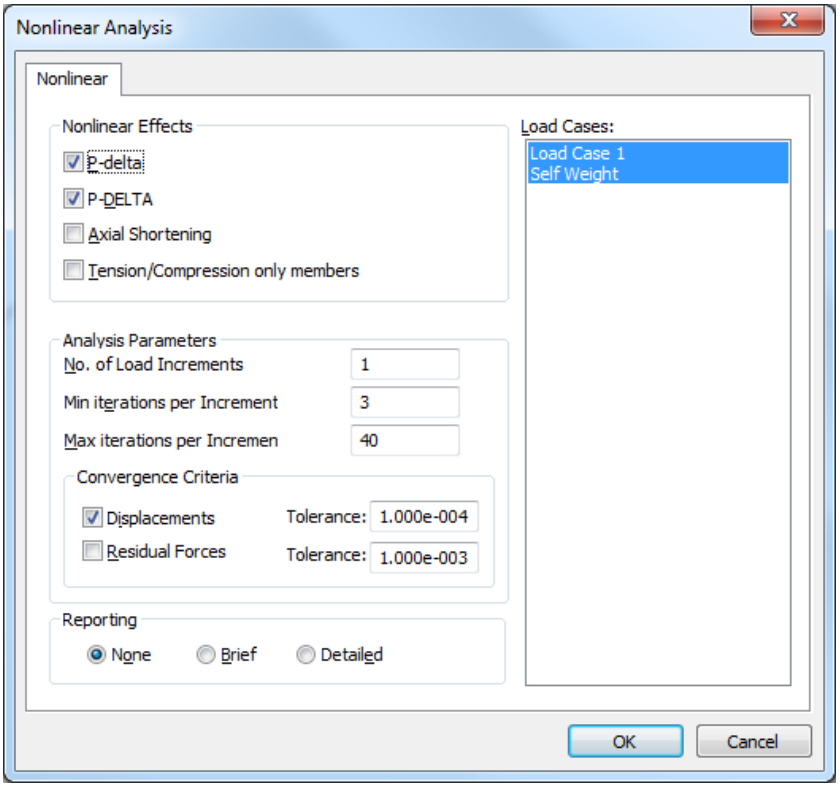
To perform a nonlinear analysis

- **Choose Analyse Nonlinear from the Analyse menu**

If this is not available, please activate the nonlinear from the Edit | Preferences | Licensing tab. If this is not possible, make sure you have installed the nonlinear analysis feature by re-running the Multiframe installer and selecting “Modify”. Also see Installing Multiframe on page 3 for more information on how to modify your analysis.

The dialog that appears allows you to:

- **Specify which nonlinear effects are to be considered in the analysis**
- **Set a number of parameters which control the solution process**
- **Set the load cases to be analysed.**



The nonlinear results for each load case may be generated separately using different nonlinear effects and analysis parameters. The setting used to generate the nonlinear results for a particular load case can easily be obtained from the Analysis Settings datasheet. Each of the nonlinear effects and analysis parameters are described below:

P-delta effect

The P-delta (P-δ) effect refers to the secondary moments induced in a member due to the axial forces within the member. The stiffness of a member is increased by tension and decreased by compression.

P-Delta effect

The P-Delta (P-Δ) effect refers to additional bending moments that are induced in a structure as it deflects. The extra moments are produced by the relocation of the loads which have moved relative to the original structure. This is modelled in Multiframe using a Finite Displacement formulation. This represents the change in geometry of a member by accounting for the rotation of the member from its original position.

Axial Shortening

Axial shortening or flexural shortening models the reduction in the length of a member due to the curvature induced by flexural bending. In most cases the effect of flexural shortening is small however, in members with large bending moments, it may be desirable to model the change in the length of the member. The implementation in Multiframe only considers shortening of the member due to the moments acting at the ends of the member. No account is made for the effect of loads applied to the member.

Tension and Compression only members

Tension only and compression only members can be considered by a nonlinear analyses. These types of members are removed or reinstated to the structure at the end of each iteration based upon the axial deformation of the member.

Analysis Parameters

Nonlinear analysis in Multiframe is performed using a Newton-Raphson solution scheme. The user may control the solution process by adjusting the number of load increments, the maximum and minimum number of iterations, the convergence tolerance and the type of convergence norm.

No. of Load Increments

The number of steps in which the loads are applied to the structure. For most analyses it is sufficient to use a single load increment.

Maximum iterations per increment

The maximum number of iterations to be performed in a single load increment. This value is set to avoid analyses that may not converge.

Minimum iterations per increment

The minimum number of iterations to be performed in a single load increment. Unless analysis time is an issue we suggest using a minimum of 3 iterations to ensure that the solution has converged. This is particularly important in tension only/compression only analyses where several iteration are often necessary to obtain a reliable solution.

Convergence Tolerance

The solver will continue to iterate until the selected converge norm is less than the Convergence Tolerance. A smaller tolerance produces a more accurate solution but will take longer to complete the analysis.

And more, with the new dialog, users are allowed to set different convergence tolerances for residual displacement and unbalanced forces. A tight tolerance for displacement and a slightly loose tolerance for force will normally get a satisfied solution as users require.

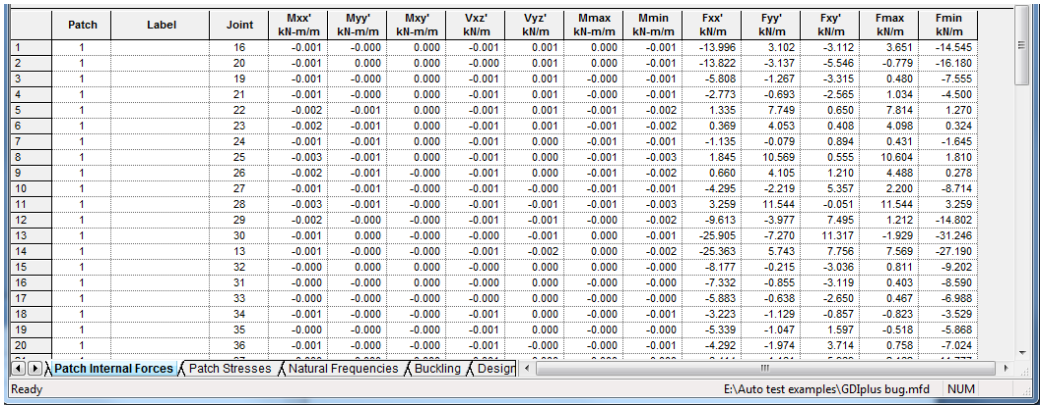
The analysis reports show the max residual displacement & force at each iteration. This could help users to determine proper convergence tolerances for their design model or even for each load case when the nonlinear analysis doesn't converge.

Type of Convergence Norm

Convergence of the solution within each increment is tested using a Residual Force or Displacement. The Residual Force norm is computed by dividing the maximum unbalanced force at the end of an iteration by the maximum force applied in the current increment. The Displacement norm is determined as the maximum change in displacement in an iteration which is normalised by the maximum total displacement at any joint in the structure. The user may choose whether to use a either or both of these norms to test convergence. Unless analysis time is an issue we suggest that both norms be used to test convergence as it is desirable for the value of both norms to be small at the end of an analysis.

Also see: Patch Internal Forces
To display the patch internal forces

- **Choose Patch Internal Forces from the Results sub-menu under the Display menu**



	Patch	Label	Joint	Mxx' kN-m/m	Myy' kN-m/m	Mxy' kN-m/m	Vxz' kN/m	Vyz' kN/m	Mmax kN-m/m	Mmin kN-m/m	Fxx' kN/m	Fyy' kN/m	Fxy' kN/m	Fmax kN/m	Fmin kN/m
1	1		16	-0.001	-0.000	0.000	-0.001	0.001	0.000	-0.001	-13.996	3.102	-3.112	3.651	-14.545
2	1		20	-0.001	0.000	0.000	-0.000	0.001	0.000	-0.001	-13.822	-3.137	-5.546	-0.779	-16.180
3	1		19	-0.001	-0.000	0.000	-0.001	0.001	-0.000	-0.001	-5.808	-1.267	-3.315	0.480	-7.555
4	1		21	-0.001	-0.000	0.000	-0.001	0.001	-0.000	-0.001	-2.773	-0.693	-2.565	1.034	-4.500
5	1		22	-0.002	-0.001	0.000	-0.001	0.001	-0.001	-0.002	1.335	7.749	0.650	7.814	1.270
6	1		23	-0.002	-0.001	0.000	-0.001	0.001	-0.001	-0.002	0.369	4.053	0.408	4.098	0.324
7	1		24	-0.001	-0.001	0.000	-0.001	0.000	-0.001	-0.001	-1.135	-0.079	0.894	0.431	-1.645
8	1		25	-0.003	-0.001	0.000	-0.001	0.000	-0.001	-0.003	1.845	10.569	0.555	10.604	1.810
9	1		26	-0.002	-0.001	-0.000	-0.001	0.000	-0.001	-0.002	0.660	4.105	1.210	4.488	0.278
10	1		27	-0.001	-0.001	-0.000	-0.001	-0.000	-0.001	-0.001	-4.295	-2.219	5.357	2.200	-8.714
11	1		28	-0.003	-0.001	-0.000	-0.001	-0.001	-0.001	-0.003	3.259	11.544	-0.051	11.544	3.259
12	1		29	-0.002	-0.000	-0.000	-0.001	-0.001	-0.000	-0.002	-9.613	-3.977	7.495	1.212	-14.802
13	1		30	-0.001	0.000	-0.000	-0.000	-0.001	0.000	-0.001	-25.905	-7.270	11.317	-1.929	-31.246
14	1		13	-0.001	-0.000	-0.000	-0.001	-0.002	0.000	-0.002	-25.363	5.743	7.756	7.569	-27.190
15	1		32	-0.000	0.000	0.000	-0.000	0.000	0.000	-0.000	-8.177	-0.215	-3.036	0.811	-9.202
16	1		31	-0.000	-0.000	0.000	-0.000	0.000	-0.000	-0.000	-7.332	-0.855	-3.119	0.403	-8.590
17	1		33	-0.000	-0.000	-0.000	-0.000	0.000	-0.000	-0.000	-5.883	-0.638	-2.650	0.467	-6.988
18	1		34	-0.001	-0.000	-0.000	-0.001	0.000	-0.000	-0.001	-3.223	-1.129	-0.857	-0.823	-3.529
19	1		35	-0.000	-0.000	-0.000	-0.001	0.000	-0.000	-0.000	-5.339	-1.047	1.597	-0.518	-5.868
20	1		36	-0.001	-0.000	-0.000	-0.001	-0.000	-0.000	-0.001	-4.292	-1.974	3.714	0.758	-7.024

Patch Internal Forces | Patch Stresses | Natural Frequencies | Buckling | Design

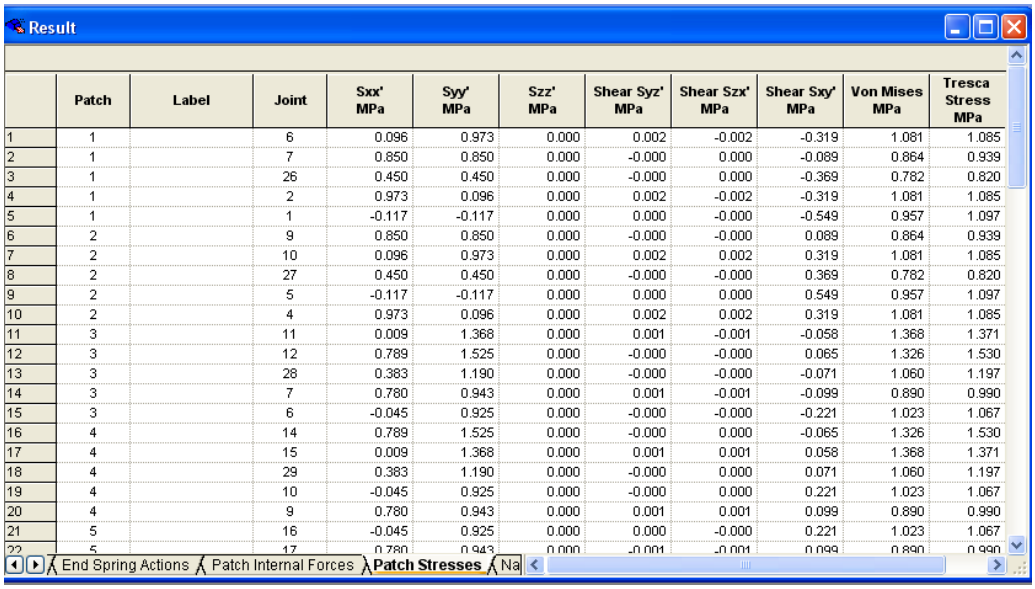
Ready E:\Auto test examples\GDIplus bug.mfd NUM

The table of patch internal forces displays the number of each patch, its label, all joint numbers within the patch and the five plate bending, transverse shear forces, plus the max & min bending moments along the principal axes, and three membrane direct forces, shear force plus max & min membrane forces along the principal axes.

Patch Stresses

To display the patch stresses

Choose Patch Stresses from the Results sub-menu under the Display menu



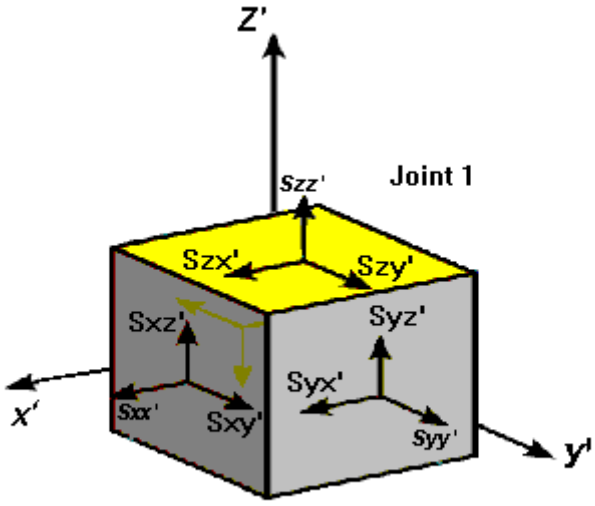
Result

	Patch	Label	Joint	Sxx' MPa	Syy' MPa	Szz' MPa	Shear Syz' MPa	Shear Szx' MPa	Shear Sxy' MPa	Von Mises MPa	Tresca Stress MPa
1	1		6	0.096	0.973	0.000	0.002	-0.002	-0.319	1.081	1.085
2	1		7	0.850	0.850	0.000	-0.000	0.000	-0.089	0.864	0.939
3	1		26	0.450	0.450	0.000	-0.000	0.000	-0.369	0.782	0.820
4	1		2	0.973	0.096	0.000	0.002	-0.002	-0.319	1.081	1.085
5	1		1	-0.117	-0.117	0.000	0.000	-0.000	-0.549	0.957	1.097
6	2		9	0.850	0.850	0.000	-0.000	-0.000	0.089	0.864	0.939
7	2		10	0.096	0.973	0.000	0.002	0.002	0.319	1.081	1.085
8	2		27	0.450	0.450	0.000	-0.000	-0.000	0.369	0.782	0.820
9	2		5	-0.117	-0.117	0.000	0.000	0.000	0.549	0.957	1.097
10	2		4	0.973	0.096	0.000	0.002	0.002	0.319	1.081	1.085
11	3		11	0.009	1.368	0.000	0.001	-0.001	-0.058	1.368	1.371
12	3		12	0.789	1.525	0.000	-0.000	-0.000	0.065	1.326	1.530
13	3		28	0.383	1.190	0.000	-0.000	-0.000	-0.071	1.060	1.197
14	3		7	0.780	0.943	0.000	0.001	-0.001	-0.099	0.890	0.990
15	3		6	-0.045	0.925	0.000	-0.000	-0.000	-0.221	1.023	1.067
16	4		14	0.789	1.525	0.000	-0.000	0.000	-0.065	1.326	1.530
17	4		15	0.009	1.368	0.000	0.001	0.001	0.058	1.368	1.371
18	4		29	0.383	1.190	0.000	-0.000	0.000	0.071	1.060	1.197
19	4		10	-0.045	0.925	0.000	-0.000	0.000	0.221	1.023	1.067
20	4		9	0.780	0.943	0.000	0.001	0.001	0.099	0.890	0.990
21	5		16	-0.045	0.925	0.000	0.000	-0.000	0.221	1.023	1.067
22	5		17	0.780	0.943	0.000	-0.001	-0.001	0.099	0.890	0.990

End Spring Actions | Patch Internal Forces | Patch Stresses | Na

The table of patch stresses is similar to the table of patch actions. It displays the number of each patch, the patch label and the joints within the patch in the leftmost columns. The remaining columns display the stresses of all joints according to the local patch coordinate system.

The patch stresses displayed in the Result window are shown in the diagram below. The stresses at all joints within its patch follow the same sign convention as the stresses at joint 1.



Transverse shear stresses follow the right hand rule i.e. if the axis is from left to right and the thumb on your right hand points in the direction towards to you.

Positive shear stress

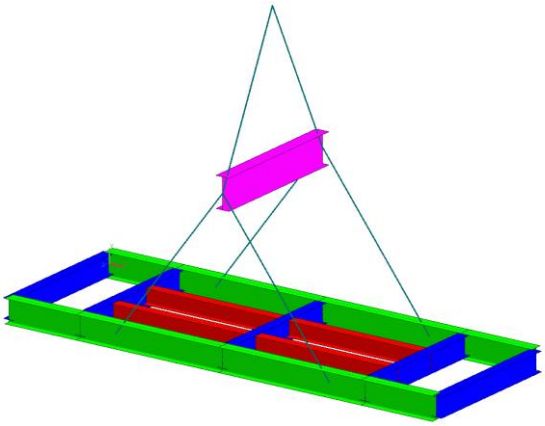
Negative shear stress

Positive axis

Nonlinear Results on page 201

Lifting Analysis

A new type of analysis has been added for analysing frames which are used for lifting calculations. Typical examples would be lifting of baskets and skids with slings which are attached to a crane.



Such models are typified by a lack of restraint in the horizontal plane which allows the frame to either rotate or swing freely. If you model a frame like this you will often get the error message “Solution does not make sense”. The lifting analysis function automatically adds additional spring restraints to the model to provide sufficient horizontal restraint to allow the model to be analysed. This eliminates the need for you to apply your own manual spring restraints.

The stiffness of these restraints is automatically set by Multiframe so as to have a minimal effect on the actions within the frame. They are set at a small proportion of the other stiffnesses in the frame. While you are creating your structural model, you should review the location of the centre of mass and ensure that the lifting point is located as near as possible to a location directly above the centre of mass of the structure and its loads.

Note also that when modelling slings or lifting cables you should release both moments and torsion at the ends of those members. If you release all of the sling members connecting to the lifting point, be sure to applied a fixed restraint to the lifting point to prevent that node from rotating.

A lifting analysis is performed via the Lifting… command in the Analysis menu. A lifting analysis is in fact a special type of linear analysis and as such the results of the lifting analysis are stored as the results of a linear analysis. If your sling members are set to Tension-Only, then a dialog will appear allowing you to take those effects into account during analysis. You are no longer required to use a NonLinear analysis for Tension-Only effects.

Some care must be taken in performing a lifting analysis as the additional spring restraints added for the analysis will also help to suppress structural mechanisms. After analysis is complete, you will notice that all nodes in the model will have reactions. You should see normal reactions at any nodes where you have applied a restraint, and at other nodes where automatic springs have been applied, you should see reactions with a small value. These small reactions are a consequence of the automatically applied springs. If you see unacceptably large reactions at any of these nodes, then you should use manual spring restraints and carry out a normal linear analysis.

Buckling Analysis

Buckling Analysis – Concepts

In Multiframe3D and Multiframe4D you can carry out a buckling analysis to determine the buckled shape of a structure with respect to a set of reference loads. This type of analysis is also referred to as a Elastic Critical Load (ECL) analysis or a stability analysis. The analysis involves the solution of a generalised eigenvalue problem of the form

$$[\mathbf{K}_e + \lambda \mathbf{K}_g]\{\mathbf{U}\} = \{\mathbf{0}\}$$

where $\mathbf{K}_e$ is the elastic stiffness matrix, $\mathbf{K}_g$ is the geometric stiffness matrix computed using a set of reference loads $\mathbf{P}_{ref}$ and λ is a load factor or multiplier. This system of equations can be solved for many values of λ (eigenvalue) and corresponding modes shapes which represent different buckling modes. The minimum load ratio that satisfies the above equation is the critical load ratio (λ_c). This gives the elastic critical load $\lambda_c \mathbf{P}_{ref}$ .It is very important to note that the reference loads define the distribution of actions within the structure, the relative distribution of these action is assumed by the analysis to remain the same at all ratios of the reference loading.

The accuracy of a buckling analysis is dependent upon the number of members used in you model. Unlike many other types of analyses, a buckling analysis requires that each component in your structure be modelled as more than one member. Indeed a good starting point would be to model each component as at least 4 members. Furthermore, analyses should also be conducted using models with even more members in order to gauge the accuracy of the analyses.

When performing a buckling analysis it is also important to accurately model restraints and secondary members that resist the buckling of members. In a linear analysis, such restraints are not important, but in a buckling analysis the out of plane buckling of a member can be significant and will be greatly affected by the existence of intermediate restraints.

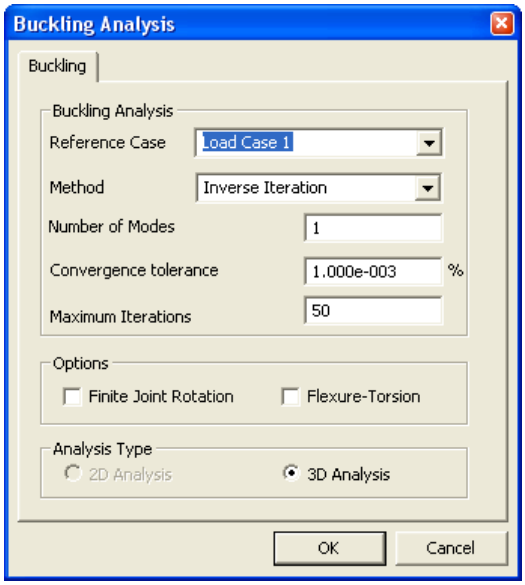
Buckling Analysis – Procedures

To perform buckling analysis:

- **Choose Buckling.. from the Analyse menu**

Buckling is only enabled when Linear or Nonlinear Analysis have been run previously.

The following dialog will appear:



Reference Case

The reference case is the case used to specify the distribution of forces though the structure. These loads will be scaled to determine the buckling modes.

Method

Two methods are provided: Inverse Iteration is the default and should be used in most cases as it is typically fastest. However, Inverse Iteration is limited to finding only the first buckling case. Where more than 1 buckling mode is required or when Inverse Iteration does not converge, you can use the Jacobi method. The Jacobi method is slower but is able to find more than a single buckling mode. As this method can be very slow it is not suitable for large models.

Number of Modes

The number of buckling modes required. This number must be between 1 and 50. Only a single buckling mode can be obtained using Inverse Iteration.

Convergence Tolerance

The minimum value, required for the 'Convergence Error'. When the convergence error falls below the ‘Convergence’ the solution has completed successfully. The convergence error is calculated at the end of each iteration when using an iterative technique such as Inverse Iteration. Typically values for the convergence error of 10-3 will give adequate results.

Maximum Iterations

The number of iterations carried out before the analysis fails. The number must be greater than 0.

Options

Multiframe 4D provides two additional options for performing buckling analysis of 3D models. When selected, these options utilise an enhanced formulation for the geometric stiffness matrix that accounts for the interaction of torsion and flexure. Full details of this formulation can be found in the text by McGuire, Gallagher and Ziemian.

Also see:

Buckling Results on page 224

Modal Analysis

Modal Analysis – Concepts

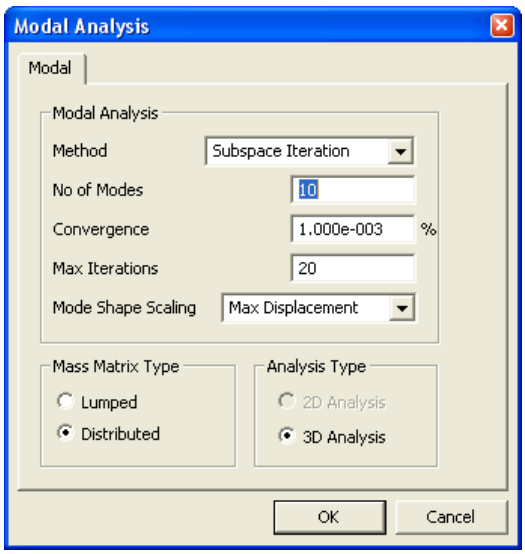
In Multiframe4D you can carry out a modal analysis of a structure and calculate up to the first 20 natural frequencies and mode shapes. Mode shapes are generated from the lowest natural frequency upwards.

Modal Analysis – Procedures

To perform Modal Analysis in Multiframe 4D

- **Select Modal from the Analyse menu**

This brings up the following dialog:



Method

Two methods are provided. Subspace iteration is the default and should be used in most cases. In cases where the subspace method does not converge, you can use the Jacobi method. This will usually be with smaller frames. The Jacobi method is slower but will find modes which are very close together.

No of Modes

The number of natural frequencies and mode shapes required. This number must be between 1 and 50.

Convergence

The minimum value, required for the 'Convergence Error'. When the convergence error falls below the 'Convergence' the solution has completed successfully. The convergence error is calculated at the end of each iteration. The convergence error is defined as

$$\text{Convergence Error} = \sum_{i=1}^{\text{Max Modes}} \left(\frac{\omega^2_{i,k+1} - \omega^2_{i,k}}{\omega^2_{i,k}} \right)^2$$

where $\omega_{i,k}$ = Natural Frequency i after iteration k.

Typically values for the convergence error from 1.0e-3 will give adequate results.

Max Iterations

The number of iterations carried out before the analysis fails. The number must be greater than 0.

Mode Shape Scaling

Specifies the technique used to scale the mode shapes. You can choose to scale mode shapes such that the maximum displacement is unity or so that the modal mass is unity.

Mass Matrix Type

When performing a dynamic and/or time history analysis the buttons Lumped and Distributed allow the user to decide on whether to use a distributed or lumped mass matrix when doing the modal analysis. Both matrices are valid so the choice is left up to the user. If you are short of memory the lumped mass matrix uses far less memory for storage. This is because the matrix is de-coupled and thus only the diagonal needs to be stored. In general, a lumped mass matrix will give slightly lower natural frequency values than an analysis using the distributed mass matrix.

Analysis Type

For most structures, the 2D Analysis button will be inactive and thus greyed out. The user will only be able to choose a 2D Analysis if either:

- **the structure (including all loads and joint displacements) are entirely in the x-y plane, or**
- **the structure is entirely in the x-y plane and only Dynamic or Seismic analysis is chosen.**

The advantages of analysing in 2D include decreased analysis time combined with decreased memory requirements, and the ability to study only the in plane natural modes when using modal analysis.

Note

If you are unsure of what values to choose, leave the current default values which will be suitable for most analyses.

Also see:

Modal Results on page 224.

Time History Analysis

Time History Analysis - Concepts

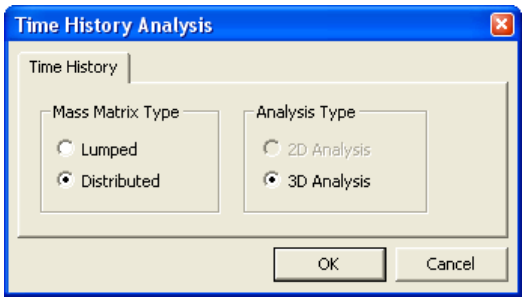
Once you have defined your dynamic loads or accelerations, you can perform a Time History analysis to compute the response of the structure at each of the time steps you have specified.

Time History Analysis - Procedures

To perform a Time History Analysis

- **Select Time History Analysis from the Analyse menu**

This command is only available after your have defined Dynamic Loads such as Time Force series or Seismic acceleration series. The following dialog will appear:



The options for Mass Matrix Type and Analysis Type are the same as for Modal Analysis, see page 192.

Batch Analysis

Batch Analysis – Concepts

As described in the Analysis section on page 183 Multiframe has a range of different analysis types. Batch Analysis allows you to run different analysis types and adjust the analysis settings in one dialog:

- **Specify which analyses you wish to run**
- **Set the analysis settings for each analysis type**

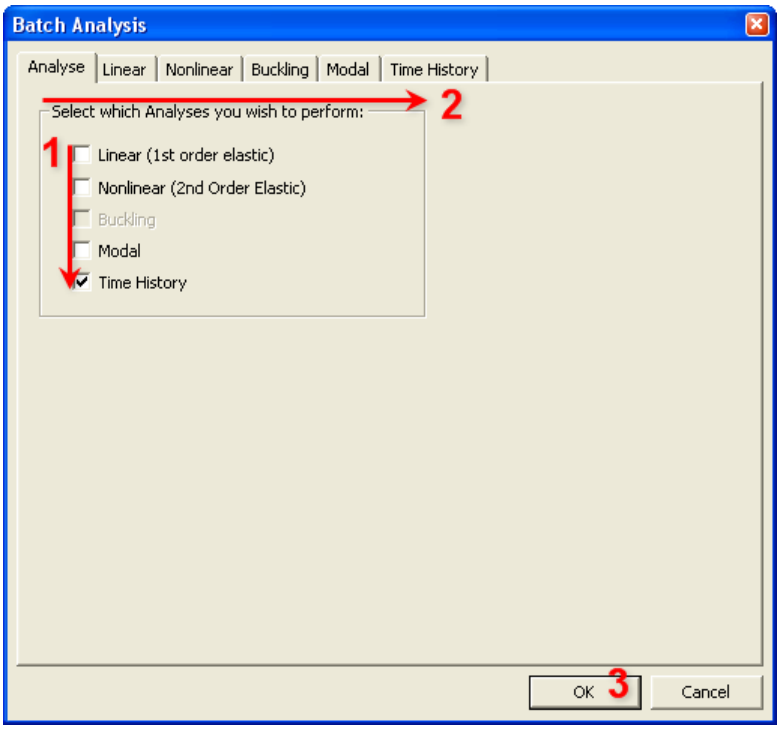
Batch Analysis is not available for Multiframe 2D.

Batch Analysis – Procedures

To perform a Batch Analysis

- **Select Batch Analysis from the Analyse menu**

This brings up the following dialog:



The dialog is designed to work in a 3 step process:

1. **Specify which analyses you wish to run**
2. **Set the analysis settings for each analysis type**
3. **Press OK**

Buckling is only enabled when either

- **Linear or Nonlinear Analysis have been selected**
- **“ “ have previously been run**

Viewing Results

Multiframe carries out a stiffness matrix analysis to determine the forces and displacements in the structure. The forces computed are Bending Moment, Shear Force, Torque and Axial Force. The corresponding stresses are also computed. The displacements computed are the displacements and rotations of the joints and the displacement along the members as they deflect. These results can be viewed in numerical form in the Result window or in graphical form in the Plot window.

Result Window

Tables of numerical results may be viewed in the Result window. Click in the Result window to bring it to the front or choose Result from the Window menu if the window is not visible. If the structure has not been analysed since you made changes to it, no numbers will be displayed in the window.

There are six tables of results that can be displayed in the Result window. These are a table of joint displacements, a table of joint reactions, a table of member actions or end forces, a table of stresses at the member ends, a table of member details and a table of dynamic results. Only one of these tables can be displayed at a time. The results can be viewed from the Results sub-menu under the Display menu to control which table is on display at any time. The current table is indicated with a check mark beside the appropriate menu item. You can also use the tabs at the bottom of the window.

The results for one load case at time can be viewed in the Result window. You can control which load case is currently on display by using the load case items at the bottom of the Case menu. The current load case is indicated with a check mark to the left of its name in the menu.

You may find the Result Layout command from the Window menu useful when viewing your results. This command arranges the Plot, Load and Result windows on the screen so that you can easily view graphical and numerical information at the same time. Remember that the Symbol… command under the View Menu will allow you to display the joint and/or member numbers on the graphics in the Plot and Load windows. This will allow you to refer to the text and graphics more conveniently.

In all of the tables drawn in the Result window, you can resize the columns by dragging the lines which separate the column titles. You can also change the text font and size used in the table by using the Font… command from the View Menu. Using the Numbers… command from the Format window can control the format of the numbers displayed in the table.

On Windows you can Right-Click on any column heading to sort or hide that column. Select and range of headings and use Right-Click to show hidden columns.

You can copy data from the Result window to the clipboard for use with other applications. To copy a single number from the table,

- **Click on the number to be copied**
- **Choose Copy from the Edit menu**

To copy a column from the table

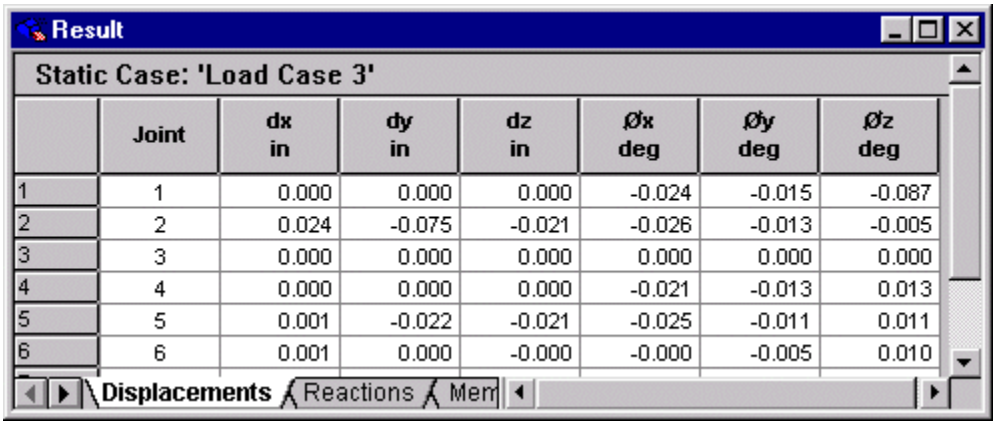
- **Click on the title of the column to select it**
- **Choose Copy from the Edit menu**

In a similar way you can select a row by clicking on the row number at the left hand end of the row or you can select the whole table by clicking in the box at the top left corner of the table.

Joint Displacements

To display the joint displacements

- **Choose Joint Displacements from the Results sub-menu under the Display menu**



	Joint	dx in	dy in	dz in	Øx deg	Øy deg	Øz deg
1	1	0.000	0.000	0.000	-0.024	-0.015	-0.087
2	2	0.024	-0.075	-0.021	-0.026	-0.013	-0.005
3	3	0.000	0.000	0.000	0.000	0.000	0.000
4	4	0.000	0.000	0.000	-0.021	-0.013	0.013
5	5	0.001	-0.022	-0.021	-0.025	-0.011	0.011
6	6	0.001	0.000	-0.000	-0.000	-0.005	0.010

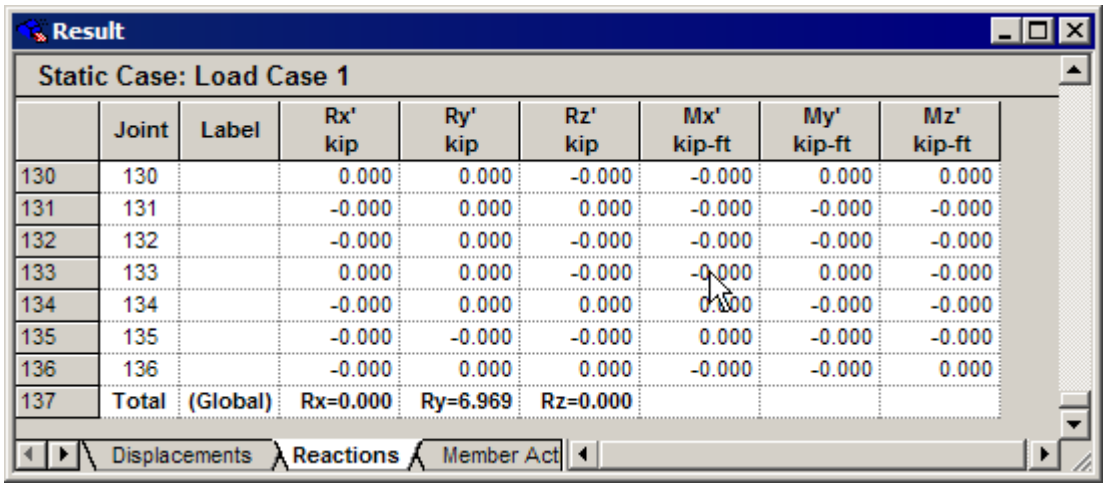
The table of joint displacements displays the number of each joint at the left of each row and the deflections and rotations in the direction of each global axis appear in the six columns. The units for each variable are shown underneath the title of the column in the table.

The displacements in this table are displayed in the directions of the local degree of freedoms at each joint. These are defined by the orientation of the joint.

Joint Reactions

To display the joint reactions

- **Choose Joint Reactions from the Results sub-menu under the Display menu**



	Joint	Label	Rx' kip	Ry' kip	Rz' kip	Mx' kip-ft	My' kip-ft	Mz' kip-ft
130	130		0.000	0.000	-0.000	-0.000	0.000	0.000
131	131		-0.000	0.000	0.000	-0.000	-0.000	-0.000
132	132		-0.000	0.000	-0.000	-0.000	-0.000	-0.000
133	133		0.000	0.000	-0.000	-0.000	0.000	-0.000
134	134		-0.000	0.000	0.000	0.000	-0.000	-0.000
135	135		-0.000	-0.000	-0.000	0.000	-0.000	-0.000
136	136		-0.000	0.000	0.000	-0.000	-0.000	0.000
137	Total	(Global)	Rx=0.000	Ry=6.969	Rz=0.000			

The table of joint reactions displays the number of each joint at the left of each row and the reactions in the direction of each global axis appear in the six columns. A sum of the three force components appears at the bottom of the table. The units for each variable are shown underneath the title of the column in the table.

The reactions in this table are displayed in the directions of the local degree of freedoms at each joint. These are defined by the orientation of the joint.

Member Actions

To display the member actions

- **Choose Member Actions from the Results sub-menu under the Display menu**

Result

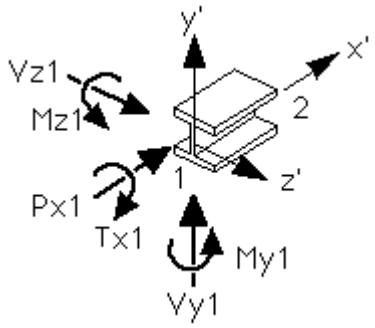
Static Case: 1.25DL + 1.5LL

	Memb	Label	Joint	Px' kip	Vy' kip	Vz' kip	Tx' kip-ft	My' kip-ft	Mz' kip-ft
13	7		7	8.784	-0.300	0.000	0.000	0.000	-92.709
14	7		8	-9.160	7.488	0.000	0.000	0.000	12.755
15	8		8	9.161	-7.486	0.000	0.000	0.000	-12.755
16	8		9	-9.342	10.924	0.000	0.000	0.000	-77.669
17	9		9	9.344	-10.922	0.000	0.000	0.000	77.669
18	9		10	-9.445	12.837	0.000	0.000	0.000	-141.285
19	10		10	9.438	-12.843	0.000	0.000	0.000	141.285
20	10		11	-9.543	14.858	0.000	0.000	0.000	-215.452
21	11		13	17.244	8.756	0.000	0.000	0.000	0.000

Reactions | Member Actions | Max Action

The table of member actions displays the number of each member, it's label and the joints in the first three columns and the six end forces in each of the six remaining columns.

The member actions displayed in the Result window are shown in the diagram below. The actions at joint 2 follow the same sign convention as the actions at joint 1. Moments follow the right hand rule i.e. if the thumb on your right hand points in the direction of the positive axis, the direction of curl of your fingers will indicate the direction of positive moment.



Maximum Actions

To display the member stresses

- **Choose Maximum Actions from the Results sub-menu under the Display menu**

Result

	Memb	Label	Section	Sign	Px' kN	Vy' kN	Vz' kN	Tx' kN-m	My' kN-m	Mz' kN-m	dy' mm	dz' mm
1	**1**		**139.7x5.0 CHS**	**+ve**	**0.000**	**0.000**	**0.000**	**0.000**	**0.000**	**0.000**	**0.000**	**0.000**
2	1		139.7x5.0 CHS	-ve	-5.725	0.000	0.000	0.000	0.000	0.000	-0.000	-0.000
3	1		139.7x5.0 CHS	abs	5.725	0.000	0.000	0.000	0.000	0.000	0.000	0.000
4	**2**		**139.7x5.0 CHS**	**+ve**	**0.000**	**0.000**	**0.000**	**0.000**	**0.000**	**0.000**	**0.000**	**0.000**
5	2		139.7x5.0 CHS	-ve	-5.792	0.000	0.000	0.000	0.000	0.000	-32.22	-4969.0
6	2		139.7x5.0 CHS	abs	5.792	0.000	0.000	0.000	0.000	0.000	32.227	4969.04
7	**3**		**139.7x5.0 CHS**	**+ve**	**0.000**	**0.000**	**0.000**	**0.000**	**0.000**	**0.000**	**0.000**	**0.000**
8	3		139.7x5.0 CHS	-ve	-0.739	0.000	0.000	0.000	0.000	0.000	-50.77	-9362.8
9	3		139.7x5.0 CHS	abs	0.739	0.000	0.000	0.000	0.000	0.000	50.772	9362.84
10	**4**		**139.7x5.0 CHS**	**+ve**	**0.000**	**0.000**	**0.000**	**0.000**	**0.000**	**0.000**	**0.000**	**0.000**
11	4		139.7x5.0 CHS	-ve	-0.039	0.000	0.000	0.000	0.000	0.000	-48.18	-10903.
12	4		139.7x5.0 CHS	abs	0.039	0.000	0.000	0.000	0.000	0.000	48.186	10903.6

Displacements | Reactions | Member Actions | Max Actions | Memb

The table of maximum member actions displays the number of each member and the member labels in the first two columns. The remaining columns display the maximum values of the six member actions and the two displacements. The maximum values for each member are displayed on three rows, each of the row displays the maximum absolute values, the maximum positive values and the maximum negative values of the actions in the member.

Member Stresses

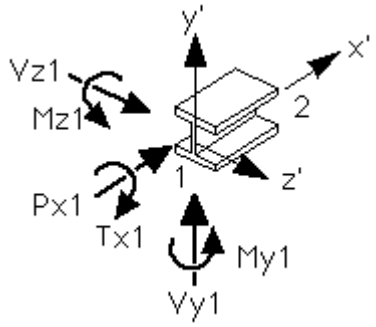
To display the member stresses

- **Choose Member Stresses from the Results sub-menu under the Display menu**

The table of member stresses is similar to the table of member actions. It displays the number of each member, the member label, and the joints at each end of the member in the leftmost columns. The remaining columns display the stresses at the ends of the member.

The table of member actions displays the number of each member and the joints in the first two columns and the six end forces in each of the six remaining columns.

The member actions displayed in the Result window are shown in the diagram below. The actions at joint 2 follow the same sign convention as the actions at joint 1. Moments follow the right hand rule i.e. if the thumb on your right hand points in the direction of the positive axis, the direction of curl of your fingers will indicate the direction of positive moment.



Maximum Stresses

To display the member stresses

- **Choose Maximum Stresses from the Results sub-menu under the Display menu**

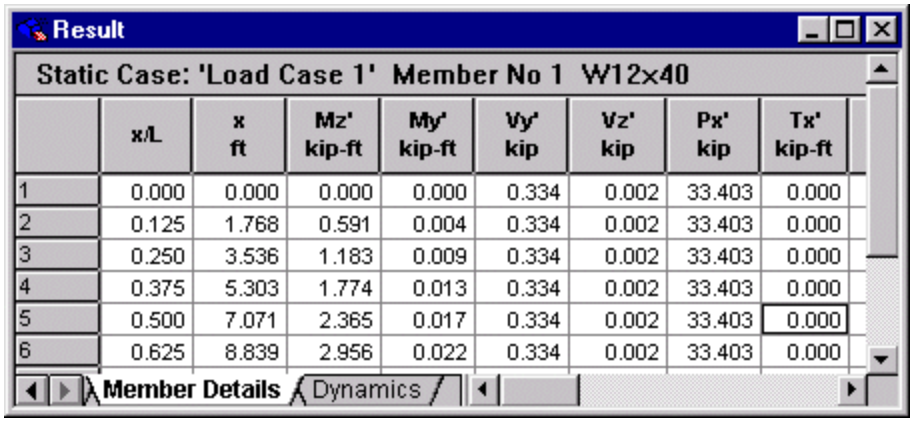
The table of maximum member stresses is similar to the table of maximum member actions. It displays the number of each member and the member label in the leftmost columns. The remaining columns display the maximum stresses within the members.

Member Details

To display a detailed table of member actions and stresses for the member currently selected in the Plot Window

- **Double click on the member to display its local diagrams**

- **Choose Member Details from the Result sub-menu under the Display menu**



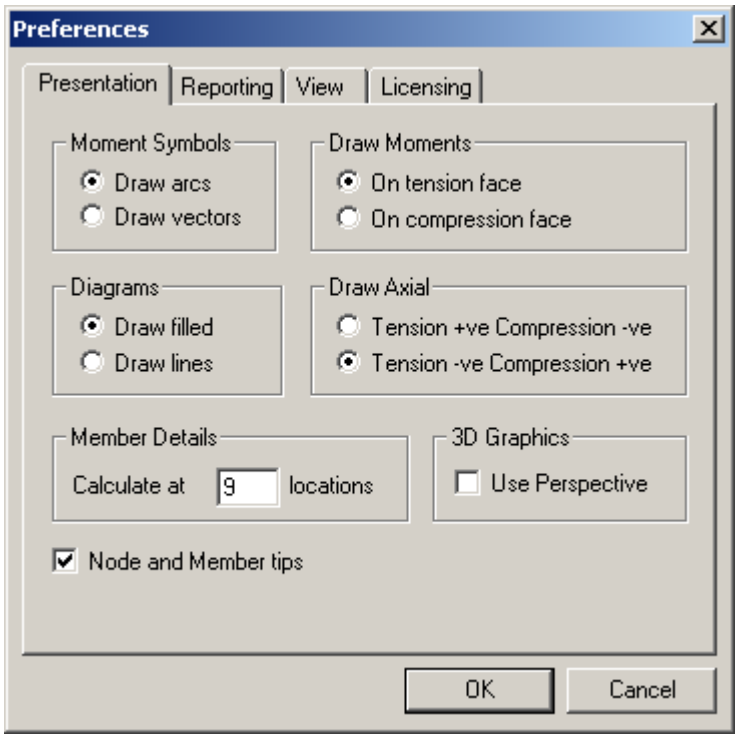
	x/L	x ft	Mz' kip-ft	My' kip-ft	Vy' kip	Vz' kip	Px' kip	Tx' kip-ft
1	0.000	0.000	0.000	0.000	0.334	0.002	33.403	0.000
2	0.125	1.768	0.591	0.004	0.334	0.002	33.403	0.000
3	0.250	3.536	1.183	0.009	0.334	0.002	33.403	0.000
4	0.375	5.303	1.774	0.013	0.334	0.002	33.403	0.000
5	0.500	7.071	2.365	0.017	0.334	0.002	33.403	0.000
6	0.625	8.839	2.956	0.022	0.334	0.002	33.403	0.000

This table gives actions and stresses at a number of points along a member. The table is displayed for the current member you have clicked on in the Plot window. The signs of the values displayed correspond with the diagram of the member and is controlled using the Preferences item (see Plot Window, Sign Convention below).

The actions, stresses and deflections are initially displayed at a number of evenly spaced locations. You can vary both the number of points and the position of each point.

To specify how many points the details will be displayed at

- **Choose Preferences from the Edit menu**



- **Type in the number of points you want to display detailed information**
- **Click the OK button**

The number of locations must be between 2 and 64.

You can also change the location of any intermediate point by clicking in the first or second column in the table and typing in a new distance or a proportional distance respectively. The first column in the table displays the distance of the point from joint 1 of the member as a proportion of the length of the member. For example, a value of 0.333 indicates the point is approximately one third of the distance along the member. The second column in the table shows the actual distance of the point from joint 1.

To change the position of a point as a proportion of the member's length

- **Click on the number in the first column of the point's row**
- **Type in a new proportion of the distance**
- **Press the Enter key**

The row's values will be re-calculated and re-displayed to show the values at the new location.

To change the distance of a point from the left hand end of the member

- **Click on the number in the second column of the point's row**
- **Type in a new distance of the point from the left hand joint**
- **Press the Enter key**

The row's values will be re-calculated and re-displayed to show the values at the new location.

Spring Actions

To display a table of the actions in spring member.

- **Choose Spring Actions from the Results sub-menu under the Display menu**

The table of spring member actions displays the number of each member, the member labels and the members joints in the first three columns. The remaining columns display the values of the six member actions in the spring members.

End Spring Actions

To display a table of the actions in member end springs.

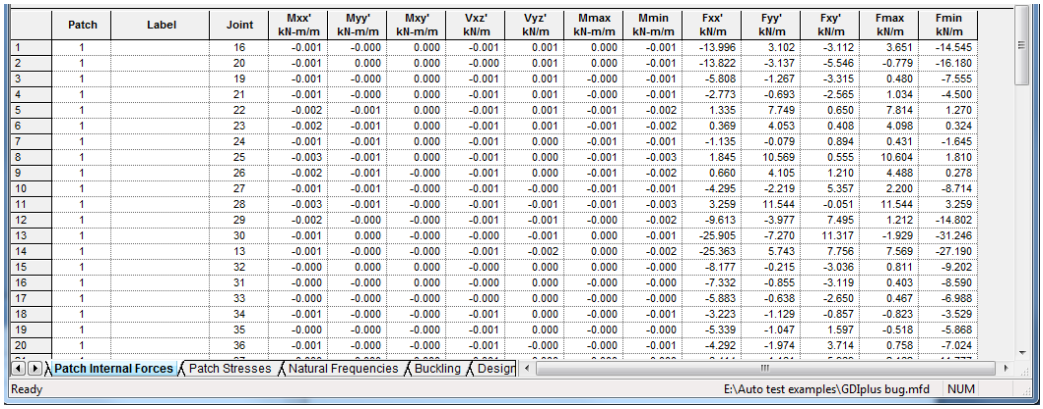
- **Choose End Spring Actions from the Results sub-menu under the Display menu**

For each end spring in the model the first three columns in the table of end spring actions displays the number of member to which the spring is attached, the member’s label and the end of the member at which the spring is located. The remaining columns display the values of the six member actions in each end spring.

Patch Internal Forces

To display the patch internal forces

- **Choose Patch Internal Forces from the Results sub-menu under the Display menu**



	Patch	Label	Joint	Mxx' kN-m/m	Myy' kN-m/m	Mxy' kN-m/m	Vxz' kN/m	Vyz' kN/m	Mmax kN-m/m	Mmin kN-m/m	Fxx' kN/m	Fyy' kN/m	Fxy' kN/m	Fmax kN/m	Fmin kN/m
1	1		16	-0.001	-0.000	0.000	-0.001	0.001	0.000	-0.001	-13.996	3.102	-3.112	3.651	-14.545
2	1		20	-0.001	0.000	0.000	-0.000	0.001	0.000	-0.001	-13.822	-3.137	-5.546	-0.779	-16.180
3	1		19	-0.001	-0.000	0.000	-0.001	0.001	-0.000	-0.001	-5.808	-1.267	-3.315	0.480	-7.555
4	1		21	-0.001	-0.000	0.000	-0.001	0.001	-0.000	-0.001	-2.773	-0.693	-2.565	1.034	-4.500
5	1		22	-0.002	-0.001	0.000	-0.001	0.001	-0.001	-0.002	1.335	7.749	0.650	7.814	1.270
6	1		23	-0.002	-0.001	0.000	-0.001	0.001	-0.001	-0.002	0.369	4.053	0.408	4.098	0.324
7	1		24	-0.001	-0.001	0.000	-0.001	0.000	-0.001	-0.001	-1.135	-0.079	0.894	0.431	-1.645
8	1		25	-0.003	-0.001	0.000	-0.001	0.000	-0.001	-0.003	1.845	10.569	0.555	10.604	1.810
9	1		26	-0.002	-0.001	-0.000	-0.001	0.000	-0.001	-0.002	0.660	4.105	1.210	4.488	0.278
10	1		27	-0.001	-0.001	-0.000	-0.001	-0.000	-0.001	-0.001	-4.295	-2.219	5.357	2.200	-8.714
11	1		28	-0.003	-0.001	-0.000	-0.001	-0.001	-0.001	-0.003	3.259	11.544	-0.051	11.544	3.259
12	1		29	-0.002	-0.000	-0.000	-0.001	-0.001	-0.000	-0.002	-9.613	-3.977	7.495	1.212	-14.802
13	1		30	-0.001	0.000	-0.000	-0.000	-0.001	0.000	-0.001	-25.905	-7.270	11.317	-1.929	-31.246
14	1		13	-0.001	-0.000	-0.000	-0.001	-0.002	0.000	-0.002	-25.363	5.743	7.756	7.569	-27.190
15	1		32	-0.000	0.000	0.000	-0.000	0.000	0.000	-0.000	-8.177	-0.215	-3.036	0.811	-9.202
16	1		31	-0.000	-0.000	0.000	-0.000	0.000	-0.000	-0.000	-7.332	-0.855	-3.119	0.403	-8.590
17	1		33	-0.000	-0.000	-0.000	-0.000	0.000	-0.000	-0.000	-5.883	-0.638	-2.650	0.467	-6.988
18	1		34	-0.001	-0.000	-0.000	-0.001	0.000	-0.000	-0.001	-3.223	-1.129	-0.857	-0.823	-3.529
19	1		35	-0.000	-0.000	-0.000	-0.001	0.000	-0.000	-0.000	-5.339	-1.047	1.597	-0.518	-5.868
20	1		36	-0.001	-0.000	-0.000	-0.001	-0.000	-0.000	-0.001	-4.292	-1.974	3.714	0.758	-7.024

Patch Internal Forces | Patch Stresses | Natural Frequencies | Buckling | Design

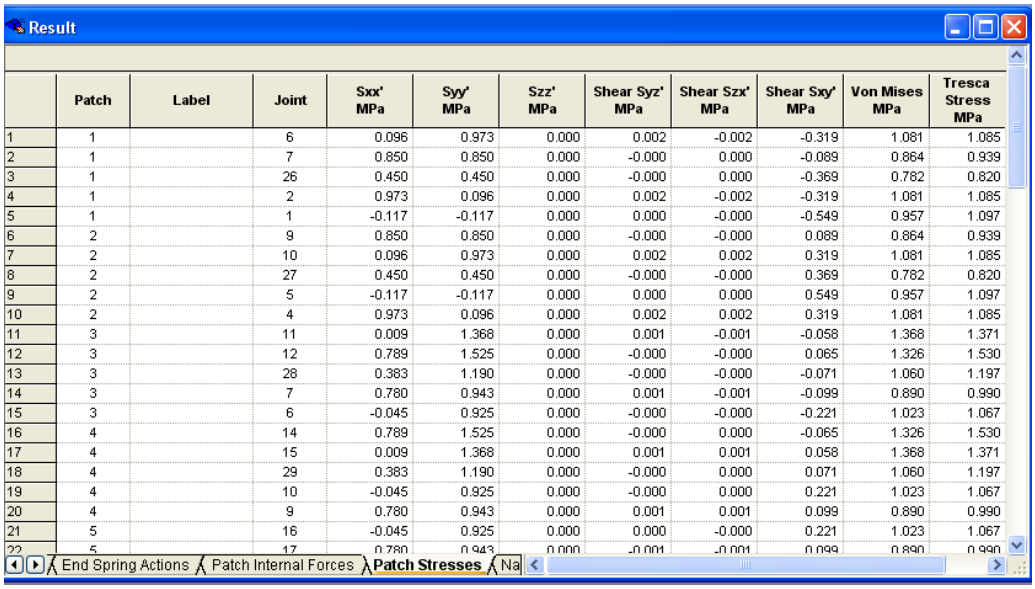
Ready E:\Auto test examples\GDIplus bug.mfd NUM

The table of patch internal forces displays the number of each patch, its label, all joint numbers within the patch and the five plate bending, transverse shear forces, plus the max & min bending moments along the principal axes, and three membrane direct forces, shear force plus max & min membrane forces along the principal axes.

Patch Stresses

To display the patch stresses

Choose Patch Stresses from the Results sub-menu under the Display menu



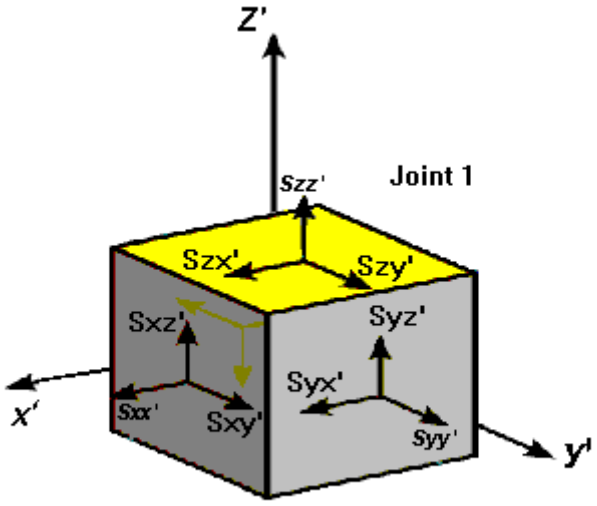
Result

	Patch	Label	Joint	Sxx' MPa	Syy' MPa	Szz' MPa	Shear Syz' MPa	Shear Szx' MPa	Shear Sxy' MPa	Von Mises MPa	Tresca Stress MPa
1	1		6	0.096	0.973	0.000	0.002	-0.002	-0.319	1.081	1.085
2	1		7	0.850	0.850	0.000	-0.000	0.000	-0.089	0.864	0.939
3	1		26	0.450	0.450	0.000	-0.000	0.000	-0.369	0.782	0.820
4	1		2	0.973	0.096	0.000	0.002	-0.002	-0.319	1.081	1.085
5	1		1	-0.117	-0.117	0.000	0.000	-0.000	-0.549	0.957	1.097
6	2		9	0.850	0.850	0.000	-0.000	-0.000	0.089	0.864	0.939
7	2		10	0.096	0.973	0.000	0.002	0.002	0.319	1.081	1.085
8	2		27	0.450	0.450	0.000	-0.000	-0.000	0.369	0.782	0.820
9	2		5	-0.117	-0.117	0.000	0.000	0.000	0.549	0.957	1.097
10	2		4	0.973	0.096	0.000	0.002	0.002	0.319	1.081	1.085
11	3		11	0.009	1.368	0.000	0.001	-0.001	-0.058	1.368	1.371
12	3		12	0.789	1.525	0.000	-0.000	-0.000	0.065	1.326	1.530
13	3		28	0.383	1.190	0.000	-0.000	-0.000	-0.071	1.060	1.197
14	3		7	0.780	0.943	0.000	0.001	-0.001	-0.099	0.890	0.990
15	3		6	-0.045	0.925	0.000	-0.000	-0.000	-0.221	1.023	1.067
16	4		14	0.789	1.525	0.000	-0.000	0.000	-0.065	1.326	1.530
17	4		15	0.009	1.368	0.000	0.001	0.001	0.058	1.368	1.371
18	4		29	0.383	1.190	0.000	-0.000	0.000	0.071	1.060	1.197
19	4		10	-0.045	0.925	0.000	-0.000	0.000	0.221	1.023	1.067
20	4		9	0.780	0.943	0.000	0.001	0.001	0.099	0.890	0.990
21	5		16	-0.045	0.925	0.000	0.000	-0.000	0.221	1.023	1.067
22	5		17	0.780	0.943	0.000	-0.001	-0.001	0.099	0.890	0.990

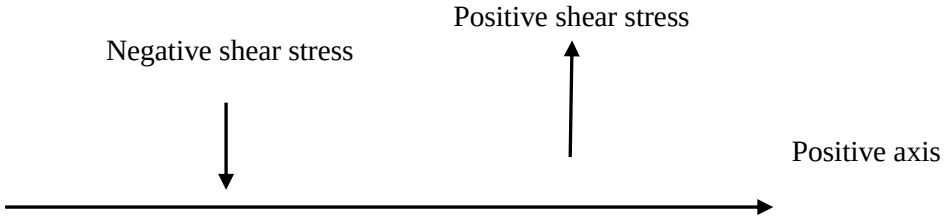
End Spring Actions | Patch Internal Forces | Patch Stresses | Na

The table of patch stresses is similar to the table of patch actions. It displays the number of each patch, the patch label and the joints within the patch in the leftmost columns. The remaining columns display the stresses of all joints according to the local patch coordinate system.

The patch stresses displayed in the Result window are shown in the diagram below. The stresses at all joints within its patch follow the same sign convention as the stresses at joint 1.



Transverse shear stresses follow the right hand rule i.e. if the axis is from left to right and the thumb on your right hand points in the direction towards to you.



Nonlinear Results

For each static load case, Multiframe stores the results of linear and nonlinear analyses separately. The user can swap between the linear and nonlinear results, for a particular load case, by either choosing the type of results from the Case menu or by clicking on the Linear/Nonlinear Results button in the Load Case toolbar.

The nonlinear results associated with a particular load case are identified by the name of the load case and an appended list of the nonlinear effects used to generate the results. The nonlinear effects are abbreviated as follows:

P-d	P-δ effect
P-D	P-Δ effect
AS	Axial shortening
TO	Tension only members
CO	Compression only members

For example, the results of a nonlinear analysis considering both the P-δ and P-Δ effects would be named as "Load Case 1 (P-d, P-D)".

This name is displayed when nonlinear results are present in the Plot and Results windows. It is also used when selecting results in the Report or Design dialogs.

Natural Frequencies

In Multiframe4D, you can display a table showing the results of a modal analysis.

- **Choose Natural Frequencies from the Result sub-menu under the Display menu**

Each row shows the frequency and period for that mode shape. In addition, the table displays the participation factors and participating mass ratios for each direction as well as the modal mass for each mode shapes. The units for each quantity are shown underneath the title of the column in the table.

Member Buckling

In Multiframe3D and Multiframe 4D, you can display a table showing the results of a buckling analysis.

- **Choose Member Buckling from the Result sub-menu under the Display menu**

Each row shows the axial force in the member and the effective length of the member for buckling of the member about both the major and minor axes of the member. In addition, the table displays the effective length factors for buckling about the major and minor axes of the member.

Time History Results:

The time history results are stored on disk in a file called "<Filename>.mth"

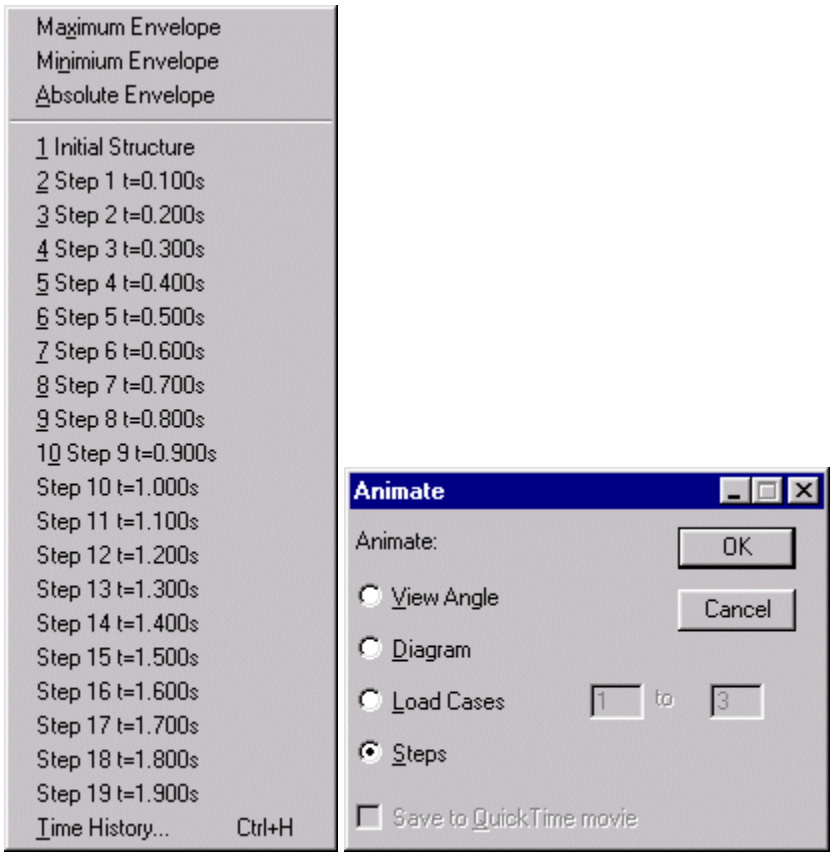
Do not delete this file while Multiframe4D is running and the associated frame is loaded.

To view the results for each time step in a dynamic or seismic load case

- **Choose the step from the Time... menu**

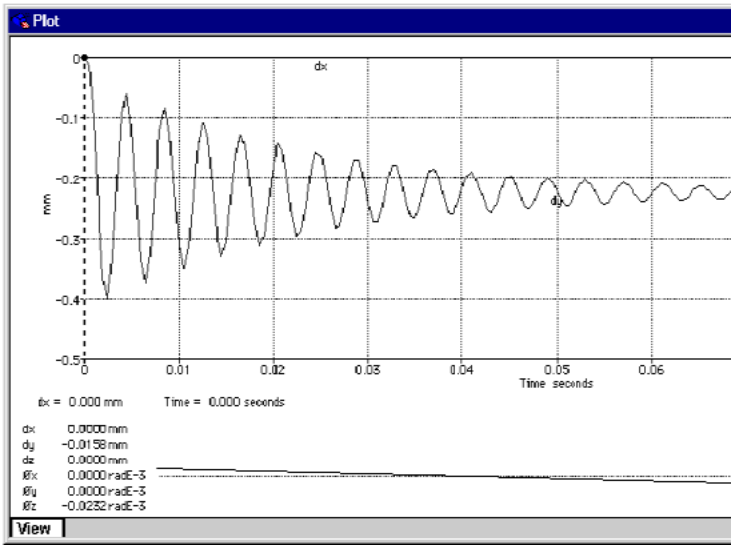
To best see the results

- **Choose Animate... from the Display menu**
- **Select Steps**



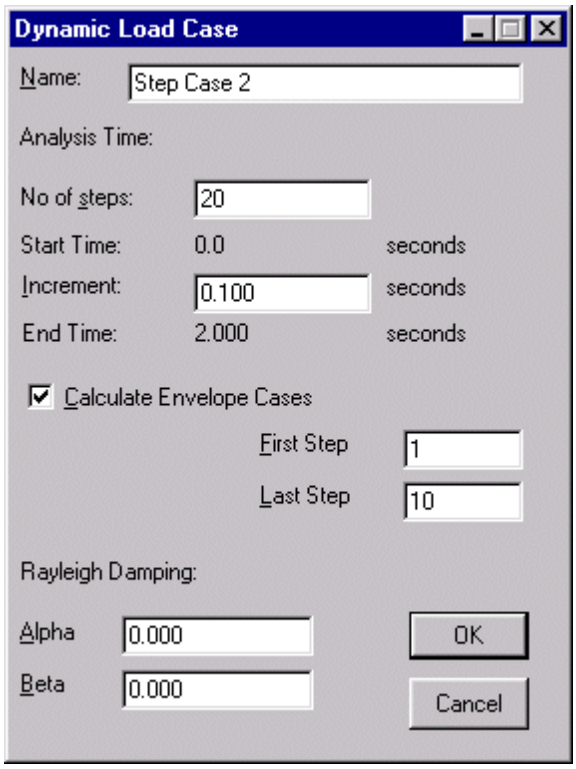
This will show you an animation including all the time steps.

When an increment in a Dynamic or Seismic load case is active you are able to view all results exactly as for a Static load case. In addition, if you select a joint in the Plot window you get a graph of the behaviour of the x y z degrees of freedom over the entire analysis.



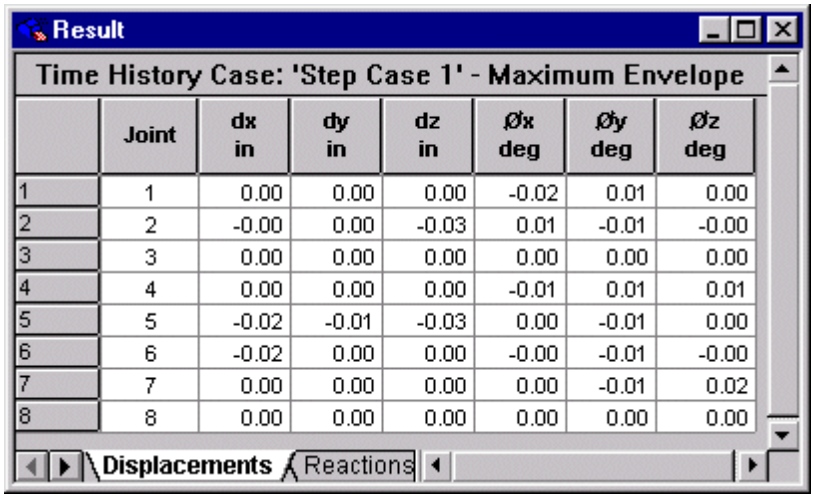
Viewing Time History Envelope Cases

Envelope cases are created for the Time History analysis if Calculate Envelope Cases check box is selected in the Dynamic or Seismic case edit dialogs. The range of cases to envelope can also be input.



The envelope cases store the absolute maximum displacement values in all cases along with the maximum, minimum and absolute values for the end forces. You can use the envelope cases to view the range of values encompassed by a dynamic or seismic analysis.

When viewing the results in the Results data window, the title bar will identify the current joint degree of freedom and the step increment that the value is associated with.



	Joint	dx in	dy in	dz in	Øx deg	Øy deg	Øz deg
1	1	0.00	0.00	0.00	-0.02	0.01	0.00
2	2	-0.00	0.00	-0.03	0.01	-0.01	-0.00
3	3	0.00	0.00	0.00	0.00	0.00	0.00
4	4	0.00	0.00	0.00	-0.01	0.01	0.01
5	5	-0.02	-0.01	-0.03	0.00	-0.01	0.00
6	6	-0.02	0.00	0.00	-0.00	-0.01	-0.00
7	7	0.00	0.00	0.00	0.00	-0.01	0.02
8	8	0.00	0.00	0.00	0.00	0.00	0.00

Plot Window

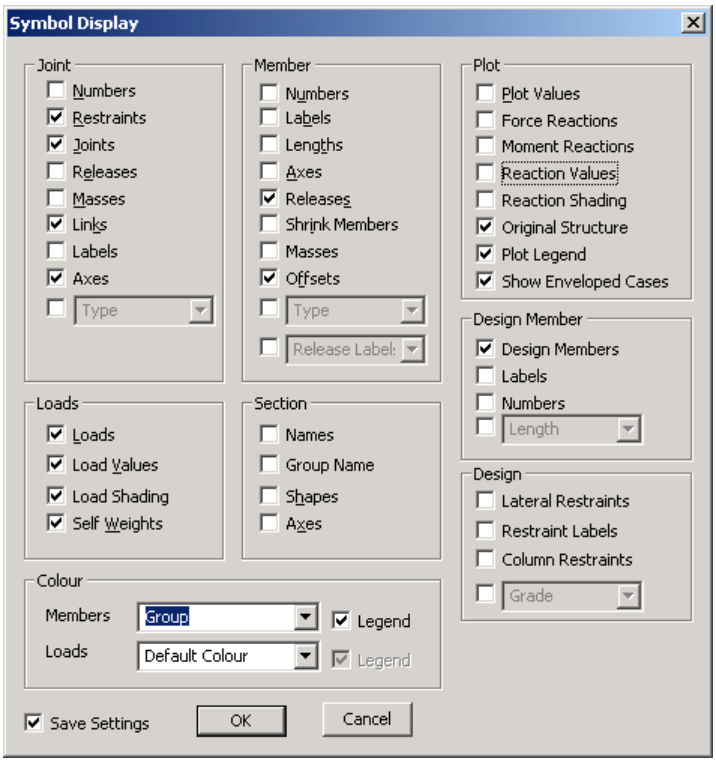
Four different types of diagrams of forces and deflections can be viewed in the Plot window. These are diagrams of the whole structure, diagrams for an individual member, reactions at a joint and deflections at a joint.

The results for one load case at time can be viewed in the Plot window. You can control which load case is currently on display by using the load case items at the bottom of the Case menu. The current load case is indicated with a check mark to the left of its name in the menu. The name of the current load case is also displayed at the bottom left hand corner of the Plot window.

When displaying diagrams of the whole structure or an individual member, you can use the items from the Diagram sub-menu under the Display menu to control which variable is to be displayed at any time. The variable currently being displayed will be indicated with a check mark to the left of the item in the menu and will also be shown at the bottom of the Plot window.

To turn on and off the display of symbols

- **Choose Symbols from the Display menu**



If you turn on the display of Plot values, the labels on the Plot diagrams for axial force and stress diagrams will have a T & C marked on the axes to indicate which is Tension and which is Compression.

The settings in this dialog can be stored and used when next starting Multiframe by choosing the Save Settings option at the bottom of the dialog.

Reactions

The reactions at joints restrained by Joint Restraints, Springs or Prescribed Displacements can be displayed in the Plot window when viewing a global diagram of the structure. The reactions are drawn as arrows with the tail of the arrow scaled according the size of the reaction. A minimum tail length is enforced so that small reactions are always visible. The reactions are displayed aligned with the local orientation of the joint and may also be drawn shaded so that smaller reactions are shown in lighter shades of colour.

The user can choose to display either or both the force and moment reactions using the Symbols dialog. A convenient shortcut button is provided in the Plot Toolbar to turn on or off the display of these reactions. The Symbols dialog also provides options for displaying the label associated with each reaction and for choosing whether to display the reactions is shades of colour.

Structure Diagrams

The diagrams of the whole structure can display bending moments, shear forces, axial forces, torque or deflection. When you have a diagram of the whole structure drawn in the Plot window and forces on display, the force diagram for each member will be superimposed on the member. With deflections on display, a diagram of the exaggerated deflection of the whole structure will be drawn. If you have performed a modal analysis and you are currently viewing a modal case, the deflected shape will represent the mode shape for that case. For all diagrams, Multiframe will choose a scale which best suits the size of your structure and the magnitude of the forces and displacements. All static load cases are drawn to the same scale. Modal results are non-dimensional.

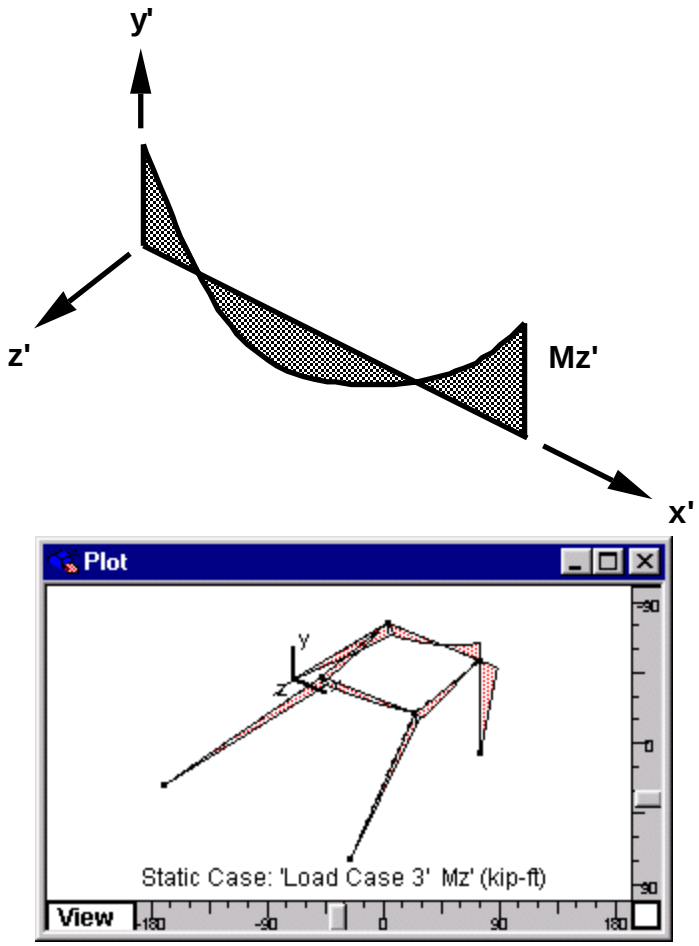
You can exaggerate the scale of a diagram

- **Choose Plot… from the Display menu**

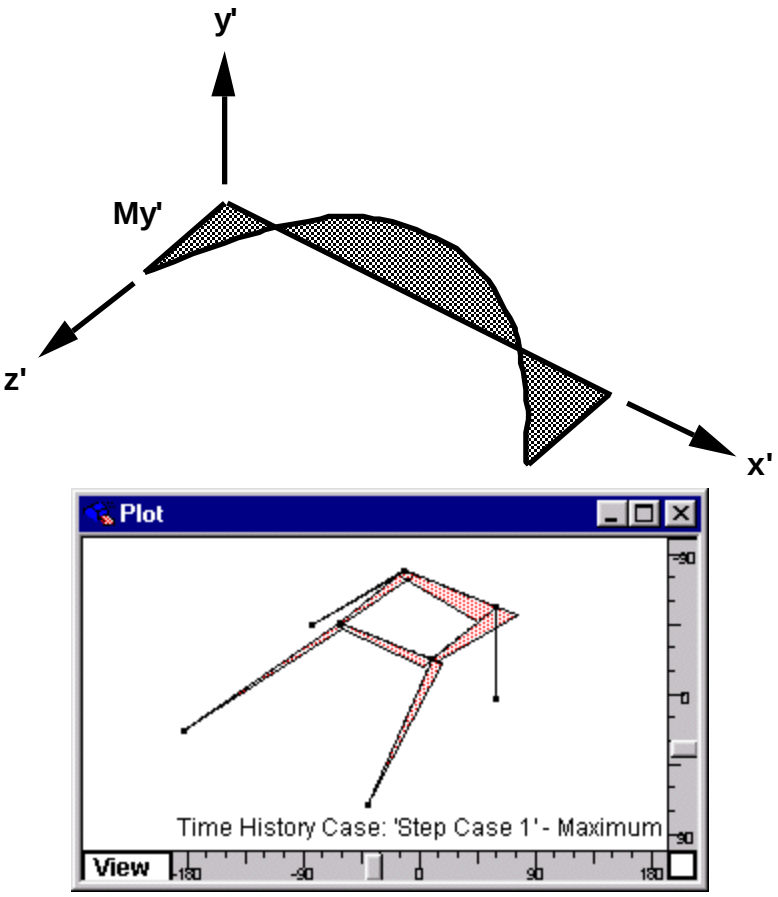
The Zoom, Shrink, Pan and Size To Fit commands can be used in the Plot window to change your view of the structure. The rotation bars can also be used to change the view point in the 3D view.

The six force diagrams that can be displayed in the Plot window, are

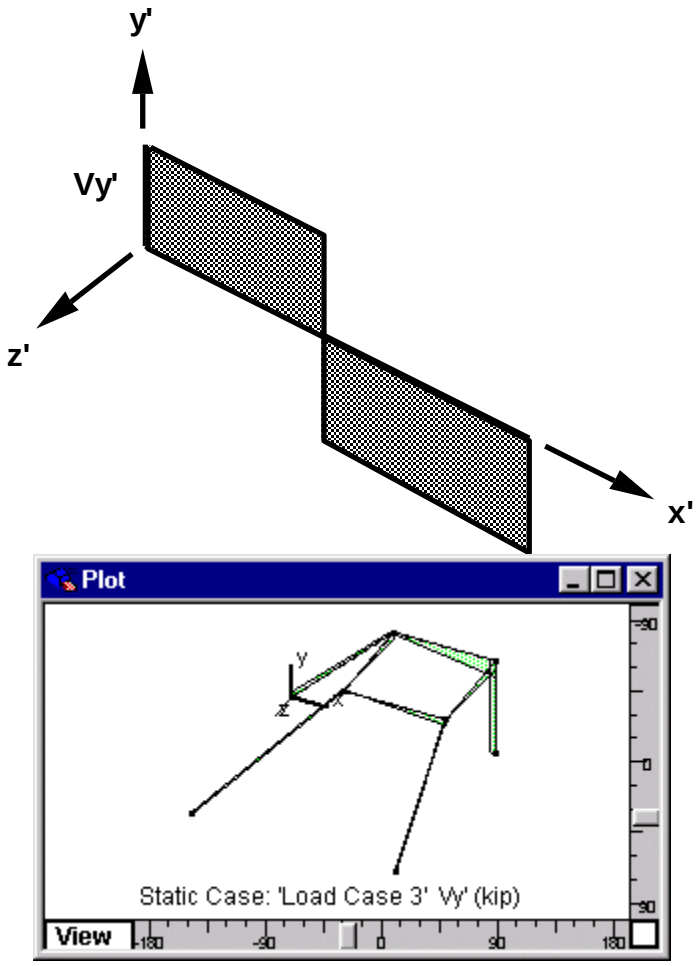
Mz' bending moments about the local z' axis of the members (in plane bending)



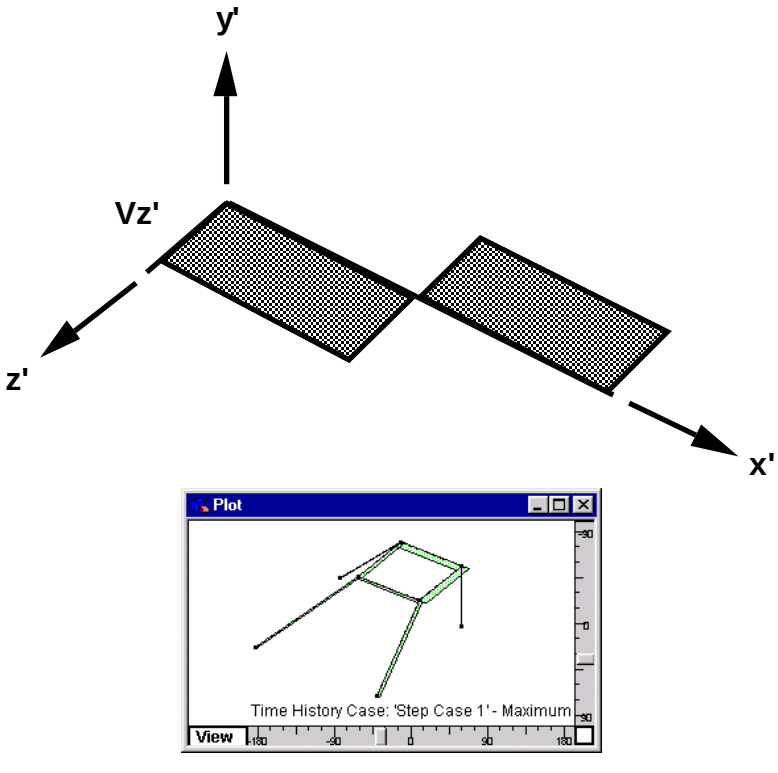
My' bending moments about the local y' axis of the members (out of plane bending)



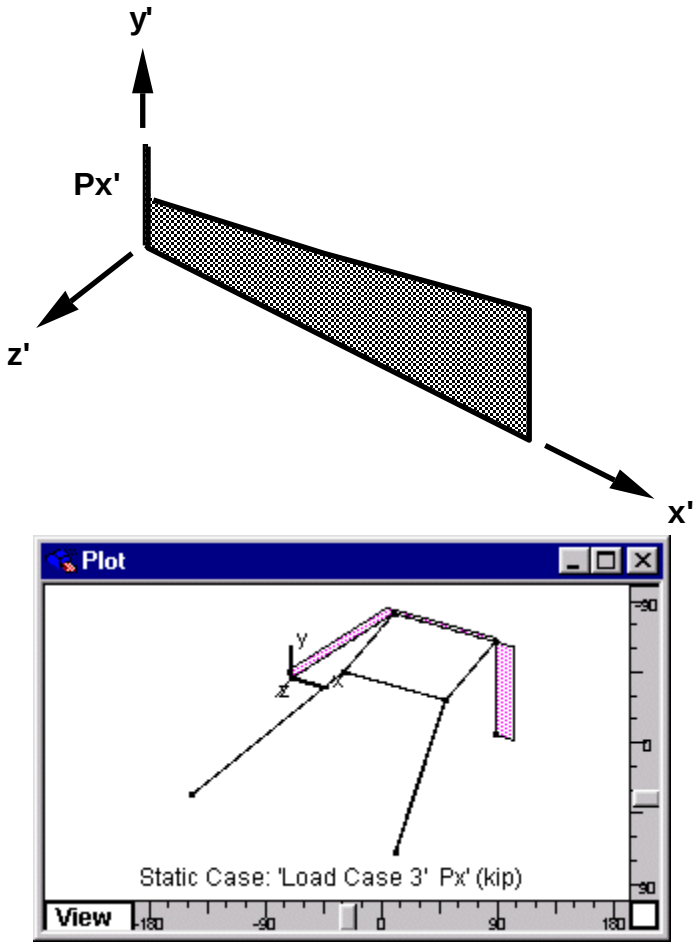
Vy' shear force through the local y' axis of the members (in plane shear)



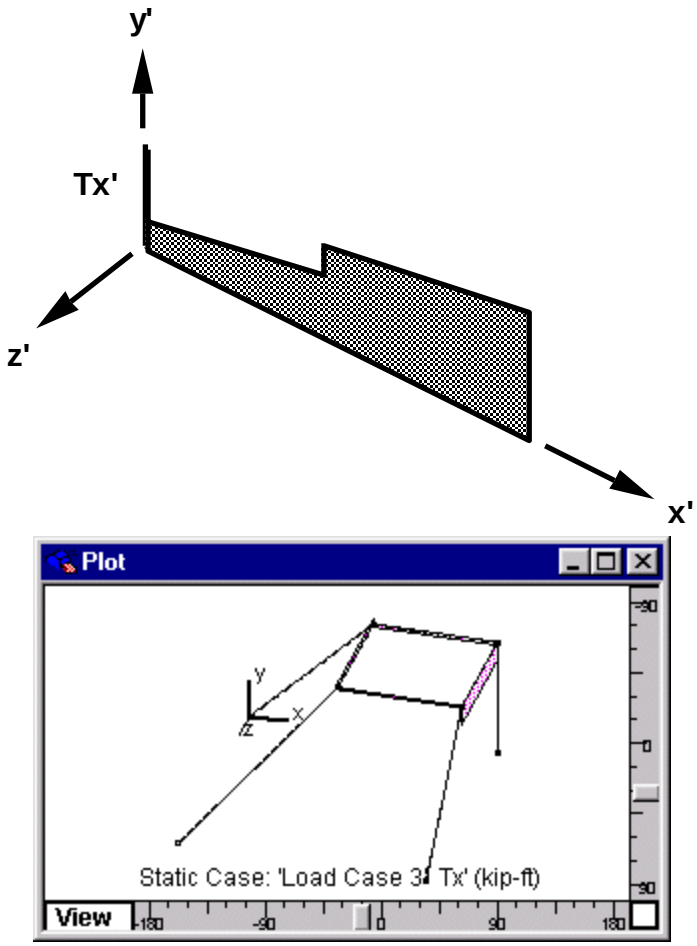
Vz' shear force through the local z' axis of the members (out of plane shear)



Px' axial force along the local x' axis of the members (tension or compression)



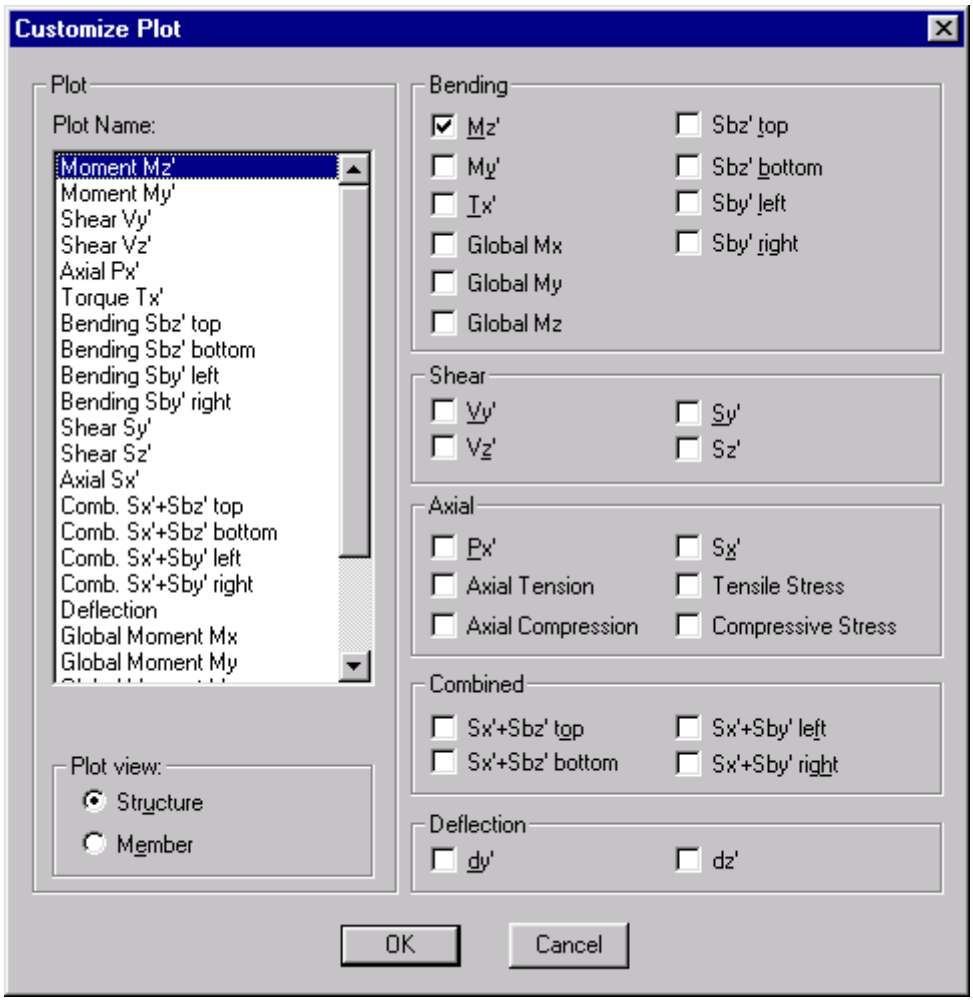
Tx'twist or torque about the local x' axis of the members



To choose which diagrams are displayed in the Plot window

- **Choose Customise Plot… from the Display menu**

A dialog will appear listing the menu items controlling the plot display and the actions, stresses and deflections which can be displayed.

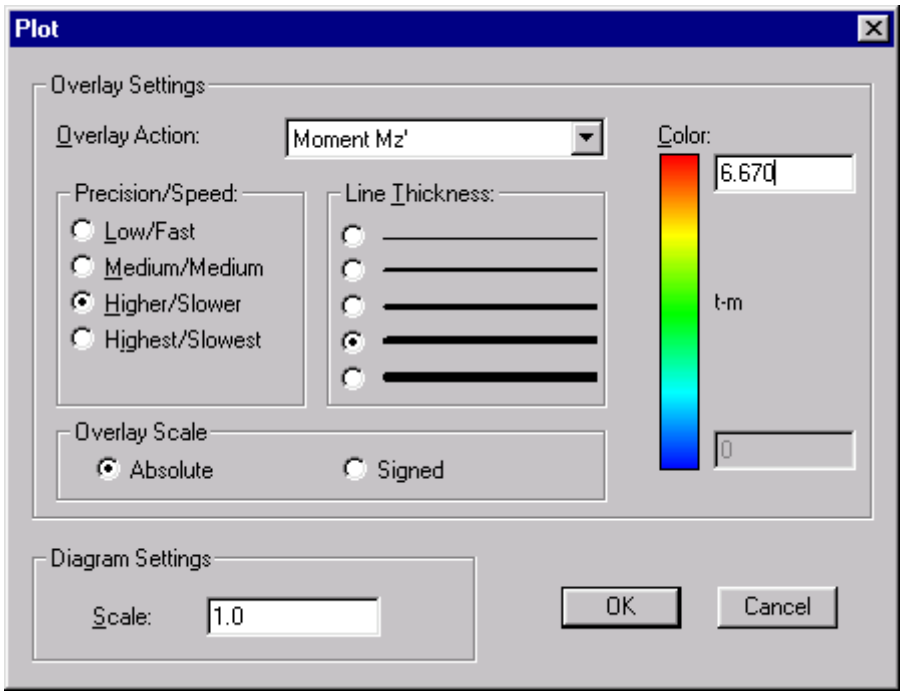


- **Click on the name of the menu item you wish to change**
- **Click on the Member button**
- **Set the check boxes of the actions you wish to display in the local member diagrams**
- **Click on the Structure button**
- **Set the check boxes of the actions you wish to display on the diagram of the whole structure**
- **Click the OK button**

You can control the values associated with the colours used when overlaying an action onto the structure deflection diagram. These colours and values are controlled using the Plot command under the Display menu.

To change the value associated with the overlaid colours

- **Choose Plot… from the Display menu**



- **Type Tab to move to the field containing the maximum threshold value**
- **Type in a new maximum threshold value to be used with red to indicate the highest actions or stresses**
- **Select the Overlay Colour Scale, either absolute or signed**
- **Click the OK button**

If the overlay is displayed using the Absolute colour scale, every part of the structure with an action of magnitude greater than or equal to the value you specified will be coloured red. Other parts of the structure will be coloured in an even gradient down to blue for those parts of the frame with an action of zero.

When the Signed overlay colour scale is selected, the overlay plot will show positive values in shades of red and negative values in shades of blue.

You can change the scale of deflected diagram plots by changing the value in the Diagram Settings/Scale field. The scale is a value relative to the default exaggeration of the diagram. Eg setting this value to 3 will result in an exaggerated plot 3 times the default scale. Setting this value to –1.0 will result in a deflection plot at true scale.

Stresses

You can display stresses as well as forces and deflections in the Plot window. Which data to display is controlled by two hierarchical menus and the deflection item under the Display menu.

To display a force diagram

- **Choose the desired item from Actions sub-menu under the Display menu**

To display a stress diagram

- **Choose the desired item from Stress sub-menu under the Display menu**

To display deflections

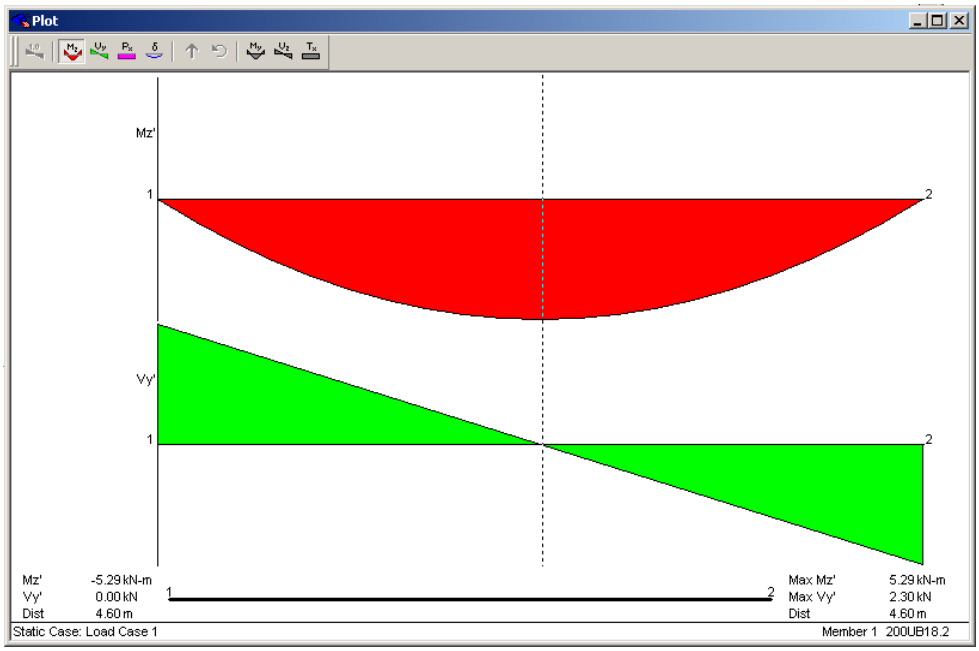
- **Choose the Deflection item from the Display menu**

You can choose to display a number of diagrams simultaneously in the Plot window. This applies to both structure and member diagrams.

Member Diagrams

The diagrams of individual members can also display bending moments, shear forces, axial forces, torque or deflections. To obtain a diagram for an individual member

- **Double click on the member you wish to view in detail**



The local diagram for that member will then be drawn in the Plot window.

The maximum and minimum values on the diagram are displayed below the graph, and below these two numbers indicate the value of the diagram at the position of the grey crosshair and the distance of that crosshair from the left hand end of the member.

You can drag the crosshair up and down the length of the member to determine the value of the diagram at any position. The crosshair will remain in the position where you release the mouse button after dragging.

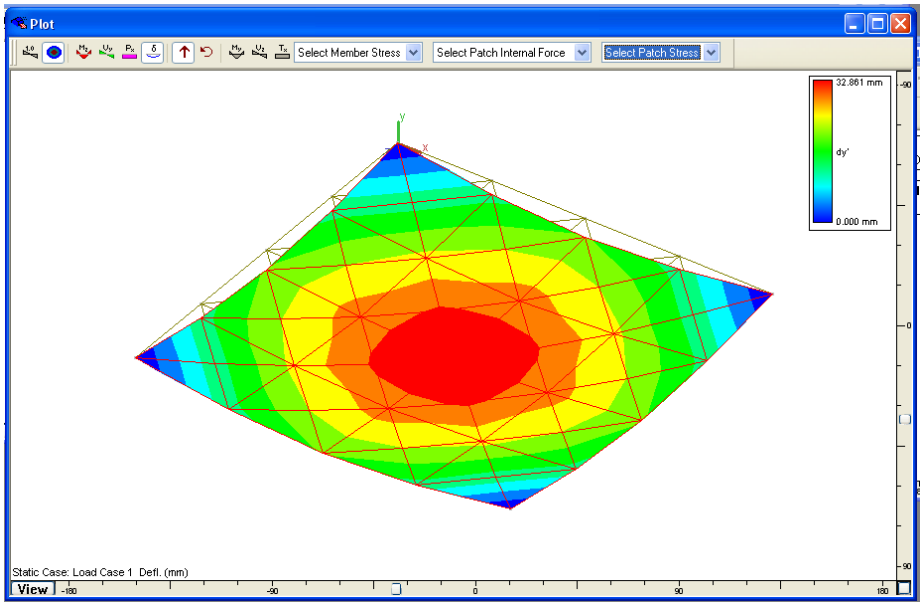
Click on the drawing at the bottom right of the window to return to a diagram of the whole structure.

Plate Diagrams

The diagrams of the plates can display bending moments, shear forces, deflection and stresses. When a diagram of the whole structure drawn in the Plot window and forces are displayed the force diagram for each plate will be superimposed on the plate. With deflections on these forces will be displayed on a diagram of the exaggerated deflection of the whole structure. All static load cases are drawn to the same scale.

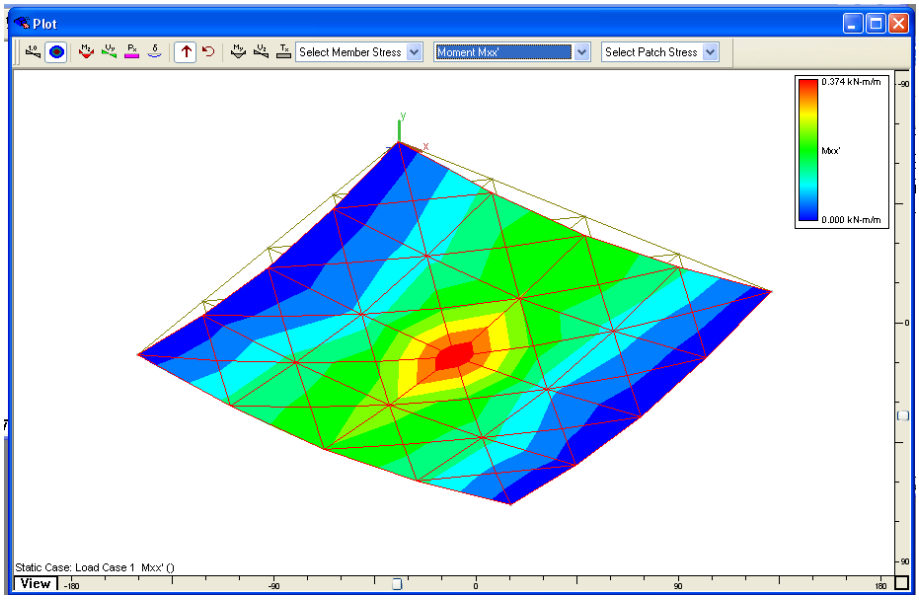
To display the contour diagram for the plate deflections

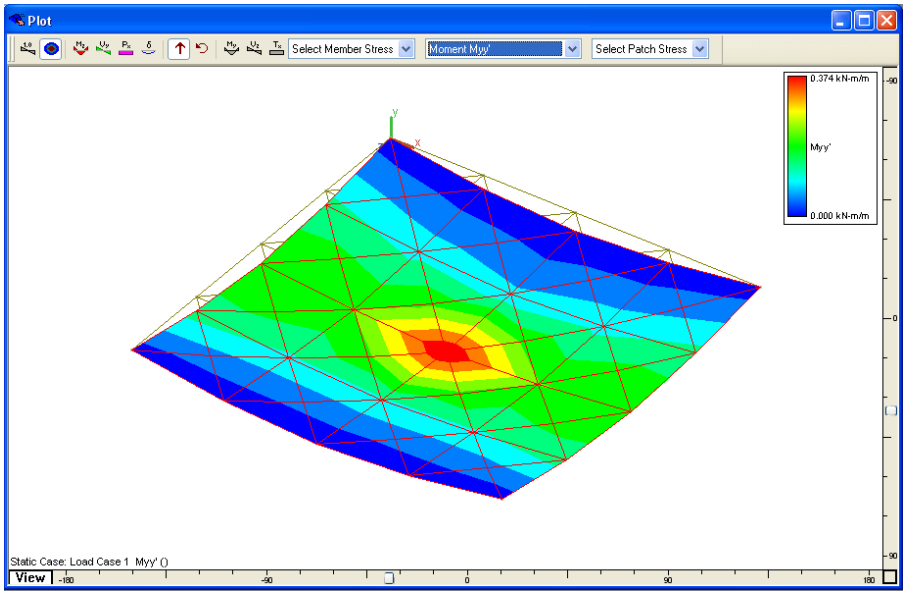
- **Choose Deflections under the Display menu**



To display the contour diagrams for the plate actions in local patch coordinate system

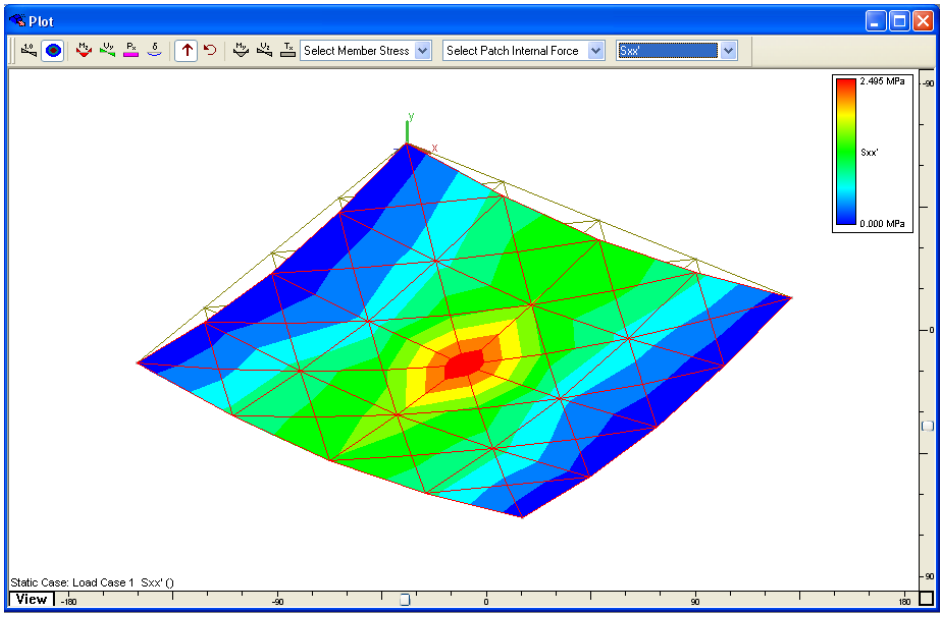
- **Choose a patch internal force from the Actions sub-menu under the Display menu**

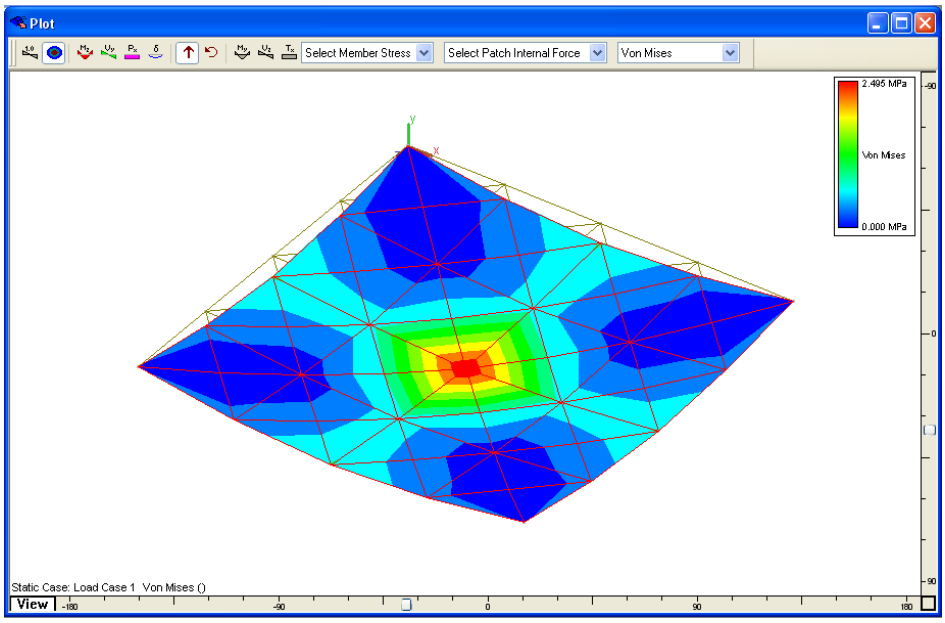




Similarly, to display the patch stress contour diagrams in local patch coordinate system

- **Choose a patch stress such as Sxx', Syy', Szz', Sxy', Syz', Szx', Von Mises or Tresca stress from the Patch Stresses sub-menu under the Display menu**



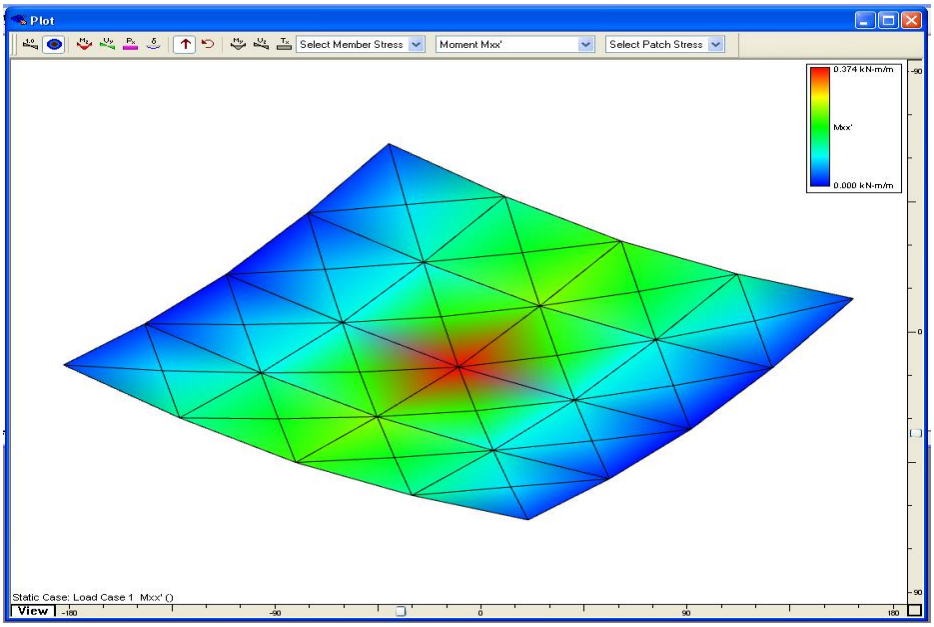


Rendering Patch Results

Multiframe allows the rendering of the display of the frame in the Frame, Load and Plot windows as an aid to visualising the relative sizes and orientation of the sections in the structure. Rendering can only be done in the 3D view in a window. All the patch diagrams such as patch bending moments, shear forces, deflection and stresses can be shown.

To render a diagram for the patch results

- **Switch to 3D view from the Current View sub-menu under the View menu**
- **Switch on rendering mode from the Display menu or Click on rendering icon**
- **Choose a patch diagram to display from the Actions or Patch Stress sub-menu under the Display menu**

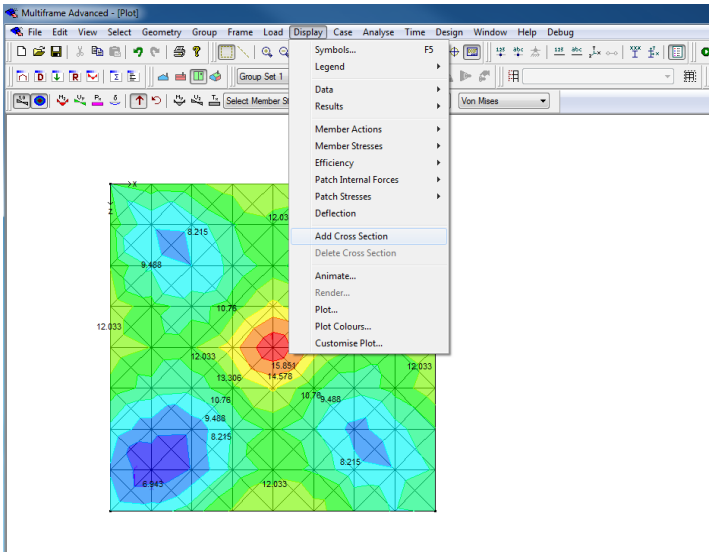


Add Patch Cross Section

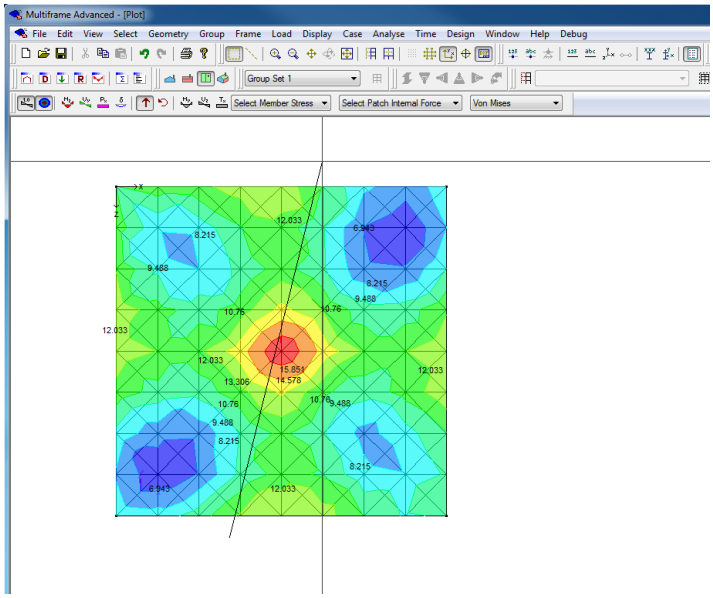
Multiframe allows patch results to be displayed on cross sections. Cross sections can be added through Plot Window after a model is analysed.

To add patch cross section

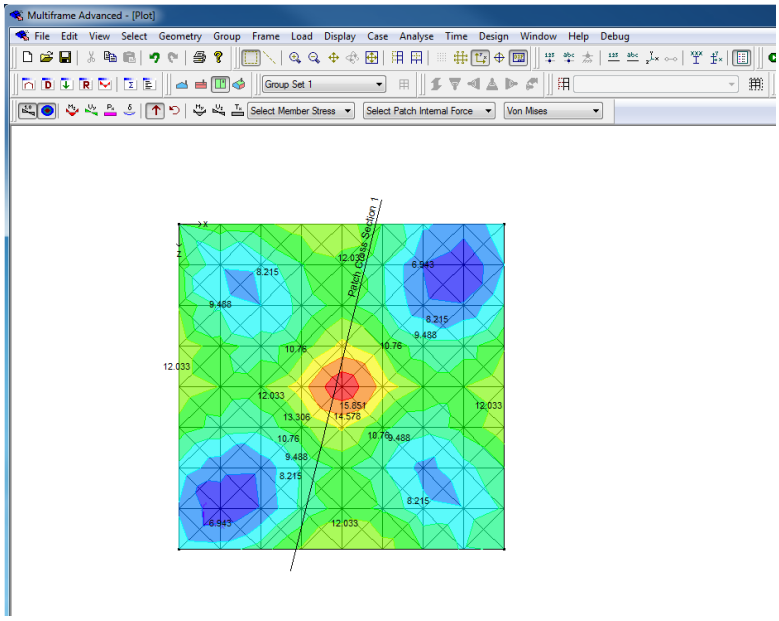
- **Switch to Plot Window**
- **Choose Add Cross Section menu from the Display menu or shortcut menu**



- **Move the pointer to the position of the first point**


Multiframe Advanced - [Plot]
File Edit View Select Geometry Group Frame Load Display Case Analyse Time Design Window Help Debug
Group Set 1
Select Member Stress
Select Patch Internal Force
Von Mises

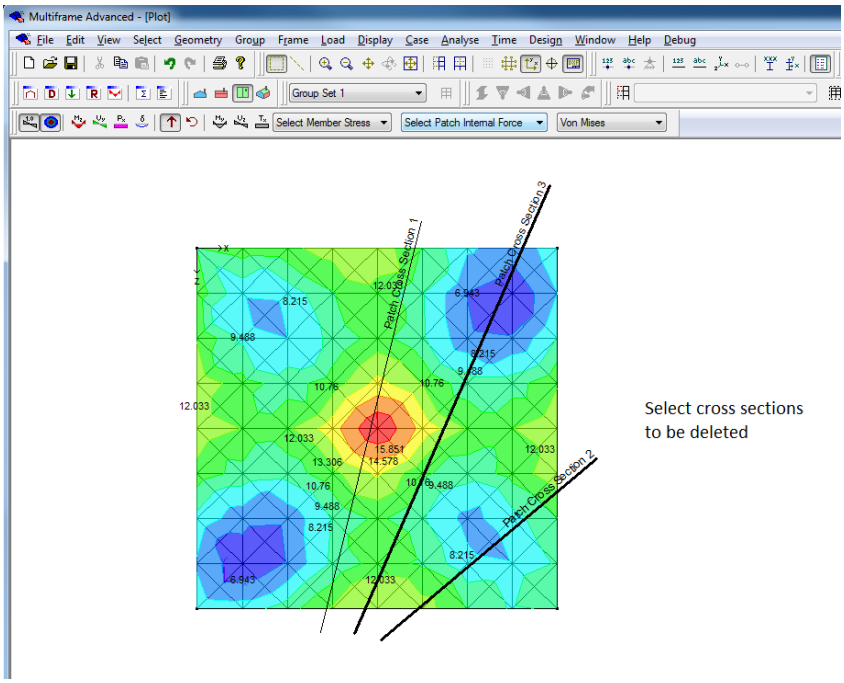
- **Click the position of the second point to complete the new Cross Section**

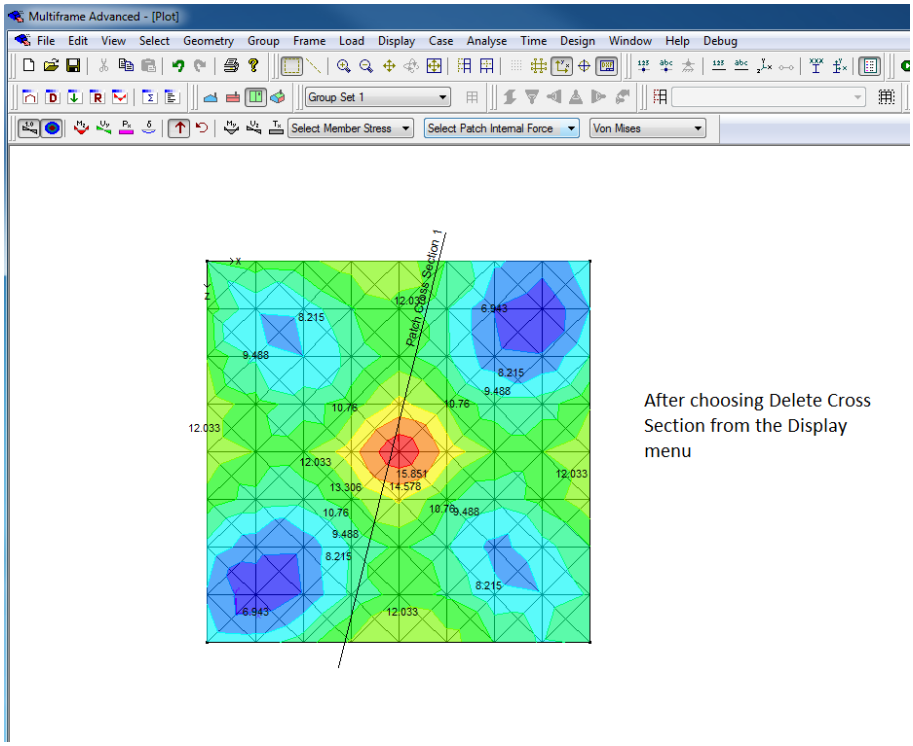

Multiframe Advanced - [Plot]
File Edit View Select Geometry Group Frame Load Display Case Analyse Time Design Window Help Debug
Group Set 1
Select Member Stress
Select Patch Internal Force
Von Mises
Patch Cross Section 1

If the grid is turned on while you draw the cross section, the position of the points will snap to the grid. The depth of the member will depend on the setting of the depth for the current view. See Depth above for an explanation of how to set this depth.

Deleting a Cross Section

- **In the Plot window, select the cross section or cross section s to be deleted**
- **Choose Delete Cross Section from the Plot menu or**
- **Press the Delete key**


Multiframe Advanced - [Plot]
File Edit View Select Geometry Group Frame Load Display Case Analyse Time Design Window Help Debug
Group Set 1
Select Member Stress
Select Patch Internal Force
Von Mises
Patch Cross Section 1
Patch Cross Section 3
Patch Cross Section 2
Select cross sections
to be deleted



After choosing Delete Cross Section from the Display menu

Envelope Case Plots

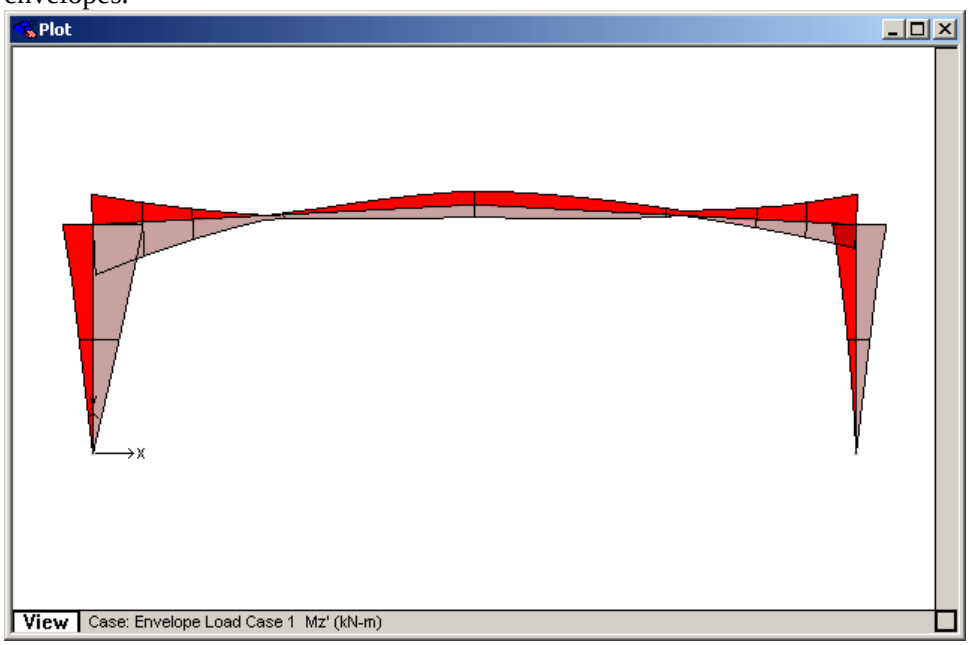
The diagrams of actions and stresses for envelope cases can be customized to display the enveloped actions in two different ways. A plot displaying only the positive and negative envelopes of the actions can be specified by

- **Choose Symbols… from the Display menu**

The symbols dialog will appear.

- **Choose Symbols… deselect the Show enveloped cases option.**
- **Click OK**

The diagrams for enveloped cases will show just the outline of the positive and negative envelopes.

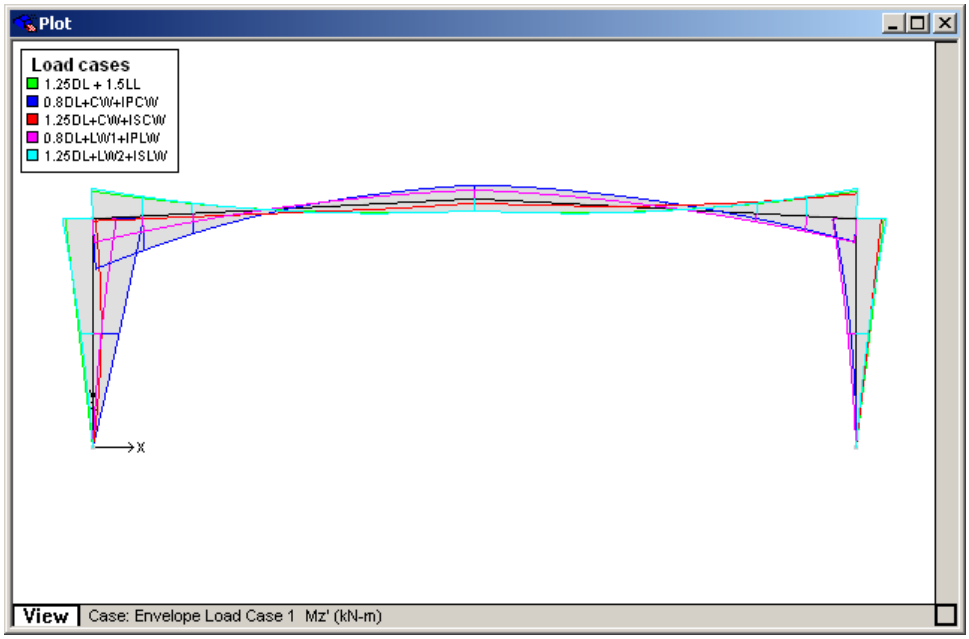


The alternative method for displaying the envelope case is to plot each of the individual load cases that are used to construct the envelope. Plots can be displayed in this way by checking the

- **Choose Symbols… from the Display menu**

The symbols dialog will appear.

- **Choose Symbols… select the Show enveloped cases option.**
- **Click OK**



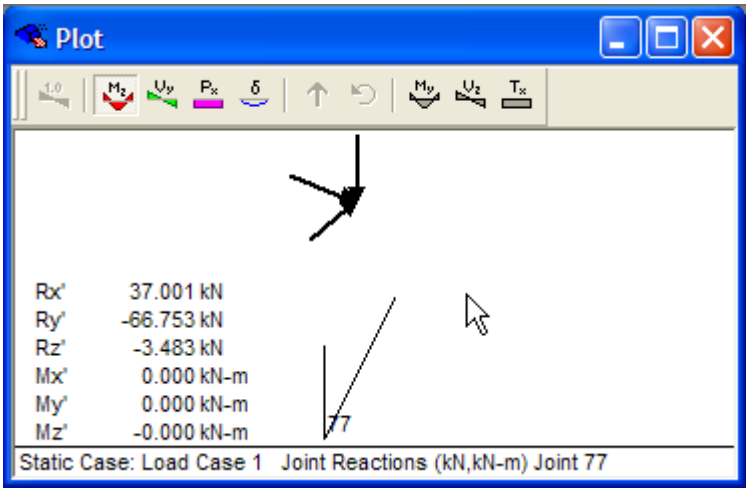
A feature of envelope load cases is that they can be edited without needing to re-analyse the model.

Joint Reactions

The diagram of a joint's reactions shows the direction and magnitudes of the reactions at the joint. Joint reactions are displayed relative to the local coordinate system of the joint.

To display the reactions for a joint

- **Ensure the moments, shear, torque or axial forces are on display**
- **Double click on the original position of the joint in the structure diagram**



The original position is marked by the point where the grey lines of the members connecting at the joint meet. The diagram of the structure will be replaced by a diagram of the joint reactions.

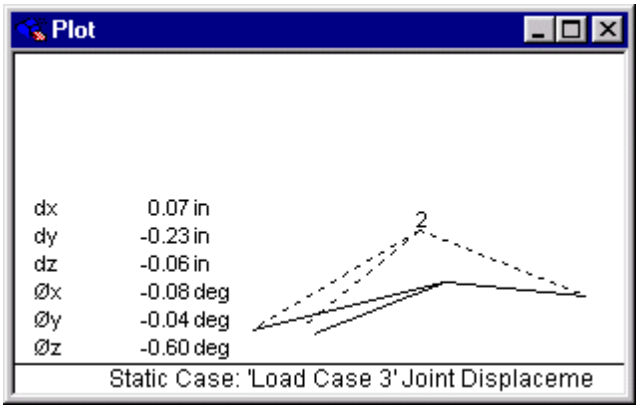
Click on the drawing at the bottom right of the window to return to a diagram of the whole structure.

Joint Displacements

The diagram of a joint's displacements indicates the vertical, horizontal and rotational displacements of the joint.

To display the displacements for a joint

- **Choose Deflection from the Display menu**
- **Double click on the original position of the joint in the structure diagram**



Click on the drawing at the bottom right of the window to return to a diagram of the whole structure.

Buckling Results

After a buckling analysis has been completed successfully, each buckling mode is stored as a separate case. The mode shapes are normalised and stored in the same way as displacements in a static analysis.

To view the load factors and mode shapes

- **Choose the buckling mode from the Buckling Modes menu in the Case menu**

The load factor is printed in the bottom left corner of the Plot window.

To view the mode shape

- **Select the Plot window**
- **Choose Deflection from the Display menu**

There are no moments, shears or other forces associated with the buckling results.

Modal Results

After a modal analysis has completed successfully, each mode is stored as a separate case. The mode shapes are normalised and stored in the same way as displacements in a static analysis.

To view the natural frequencies and mode shapes

- **Choose the mode shape from the Mode Shapes menu in the Case menu**

The frequency and period are printed in the bottom left corner of the Plot window.

To view the mode shape

- **Select the Plot window**
- **Choose Deflection from the Display menu**

There are no moments, shears or other forces associated with the modal results.

Calculations

As well as carrying out an analysis of your structure, Multiframe allows you to prepare your own design calculations by making use of the CalcSheet facility. Design calculations can be prepared and evaluated in the CalcSheet window. The calculations may be entered line by line, terminating each line by pressing return or enter. The calculations should follow the same general syntax as the Basic programming language, however, only simple calculations are supported. There is no support for looping or IF statements or comparison (<,≠, >) operators.

If the member diagram for a member is currently displayed in the Plot window, the section properties and results of analysis for the member will be automatically included as variables in the CalcSheet. These variables are listed below.

The calculations can be saved to and read from disk or printed by choosing the appropriate items from the File menu while the CalcSheet window is in front.

The following sections are available:

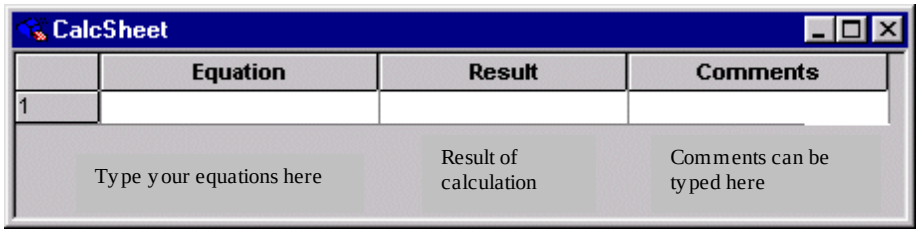
- **Calculation Sheet**
- **Pre-defined CalcSheet Variables**
- **Section Properties Variables**
- **Saving Calculations**

Calculation Sheet

The CalcSheet facility in Multiframe allows you to prepare your own design calculations and evaluate them without leaving Multiframe. The calculations can use any number of variables defined by you and can also access a number of variables from within Multiframe that contain the results of analysis and section properties for a member.

Calculations are prepared and evaluated in the CalcSheet window. To make the CalcSheet window visible

- **Choose CalcSheet from the Window menu**



The window is divided into three columns titled Equation, Value and Comment. Calculations may be entered into the equation column and when the expressions are evaluated the results will be displayed in the Value column. The Comment column may be used by you for entering any information that is relevant to the line on which it is entered. The Comment column is especially useful for entering the units for the results of calculations you carry out.

Calculations are evaluated by choosing the Calculate command from the Analyse menu. The results of the calculations are displayed in the Value column. The value shown is the value of the expression evaluated at each line.

You can the change the equations in the CalcSheet at any time and then re-evaluate the expressions using the Calculate command.

Pre-defined CalcSheet Variables

Multiframe provides you with a number of pre-defined variables for use within the CalcSheet. These variables relate to a single member in the structure at a time. You specify which member in the structure to use by clicking on that member in the Plot window and displaying its member diagram. Multiframe will automatically extract the appropriate data from that member and store it in the variables in the CalcSheet.

The pre-defined variables include the length, slope, moments, forces, displacements and section properties. Remember that these variables are only available if there is a member diagram on display in the Plot window.

A complete list of the variables is

Name	Description
dx1	x displacement of joint 1
dy1	y displacement of joint 1
dz1	z displacement of joint 1
Øx1	rotation of joint 1 about the x axis
Øy1	rotation of joint 1 about the y axis
Øz1	rotation of joint 1 about the z axis
dx2	x displacement of joint 2
dy2	y displacement of joint 2
dz2	z displacement of joint 2
dx	x' deflection at the crosshair
dy	y' deflection at the crosshair
dz	z' deflection at the crosshair
Øx2	rotation of joint 2 about the x axis
Øy2	rotation of joint 2 about the y axis
Øz2	rotation of joint 2 about the z axis
Mz	bending moment about z' at crosshair
My	bending moment about y' at crosshair
Vy	shear force through y' at crosshair
Vz	shear force through z' at crosshair
Px	axial force at crosshair
Tx	torque at crosshair
Length	Length of member
Slope	Slope of member
X1	x coordinate of joint 1
Y1	y coordinate of joint 1
Z1	z coordinate of joint 1
X2	x coordinate of joint 2
Y2	y coordinate of joint 2
Z2	z coordinate of joint 2
Dist	distance to the crosshair
MaxMz	Absolute value of the max Mz'

MaxMy	Absolute value of the max My'
MaxVy	Absolute value of the max Vy'
MaxVz	Absolute value of the max Vz'
MaxAT	Absolute value of the max axial tension
MaxAC	Absolute value of the max axial compression
MaxTx	Absolute value of the max torque
Maxdx	Absolute value of the max x' deflection
Maxdy	Absolute value of the max y' deflection
Maxdz	Absolute value of the max z' deflection
Pi	3.14159

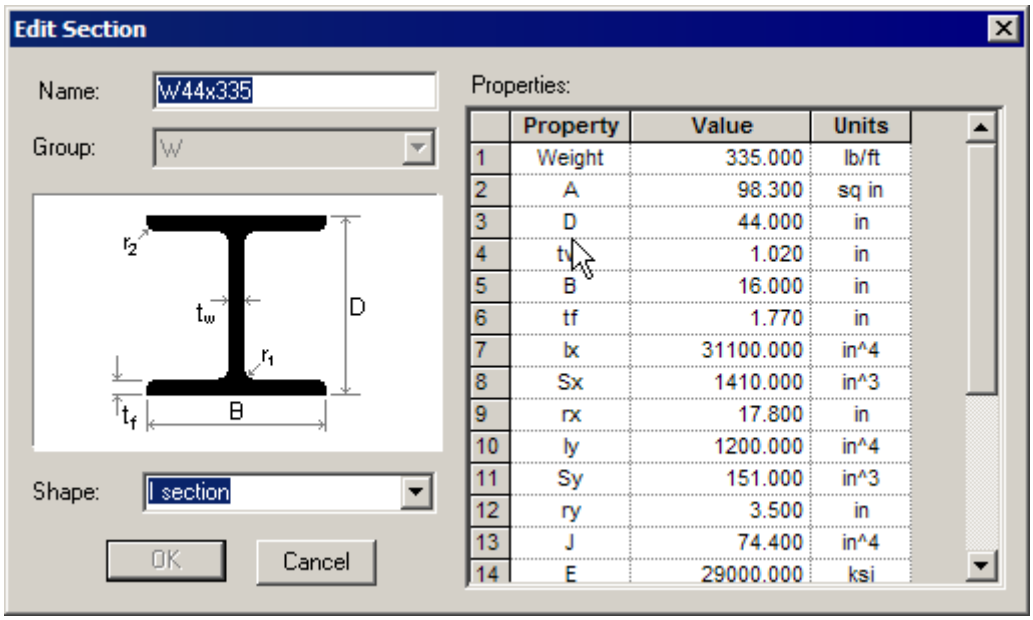
The pre-defined variables in the CalcSheet will be in the units currently specified using the Units item from the View menu (See Units below).

Section Properties Variables

Multiframe will automatically extract all the section properties for the section used for the member from the Sections Library. These properties will be stored in variables with the same names as the properties in the sections library.

To view the names of these properties

- **Choose Edit Section… from the Sections submenu under the Edit menu**



	Property	Value	Units
1	Weight	335.000	lb/ft
2	A	98.300	sq in
3	D	44.000	in
4	tw	1.020	in
5	B	16.000	in
6	tf	1.770	in
7	Ix	31100.000	in^4
8	Sx	1410.000	in^3
9	rx	17.800	in
10	Iy	1200.000	in^4
11	Sy	151.000	in^3
12	ry	3.500	in
13	J	74.400	in^4
14	E	29000.000	ksi

When you view the table of properties for a section, the name of each property will be shown in the left hand column of the table.

For example, if you wished to compute the shear stress in the web of a member you could enter a calculation such as

Shear Stress=Vy/(D*tw)

Where Vy is the shear force variable and D and tw are the section depth and web thickness taken from the properties for the section.

The constant Pi is also provided as a variable in the CalcSheet.

Saving Calculations

To save the calculations you have created

- **Ensure the CalcSheet window is in front**
- **Choose Save As from the File menu**

You can then save the calculations file on disk. The calculations in Multiframe are stored independently from the structure you are working on, so you can use the same calculations on a number of different structures.

Printing

Multiframe offers a number of options for controlling the output of data or results to a printer. Multiframe can print text, numbers and graphics.

Page Setup

Before printing data or diagrams from Multiframe, it is first necessary to set up the page size and the print quality. The Page Setup command from the File menu can be used to set up the page size you will be using in the printer. This set up need only be done once and all subsequent printing will use this format.

Setting up the Printer

First, ensure that your printer is attached to your computer with an appropriate cable and that it is switched on and has sufficient paper loaded. Refer to your computer's owner guide or your printer manual if you have any problems.

- **Choose Page Setup… from the File menu**

A dialog box will appear which allows you to choose the paper size and orientation, set the size of margins for printing and type in text to appear at the head and foot of each page of paper printed.

Select the paper size you have installed in your printer. You can also choose to enlarge or reduce the printout and adjust the output with a number of other options.

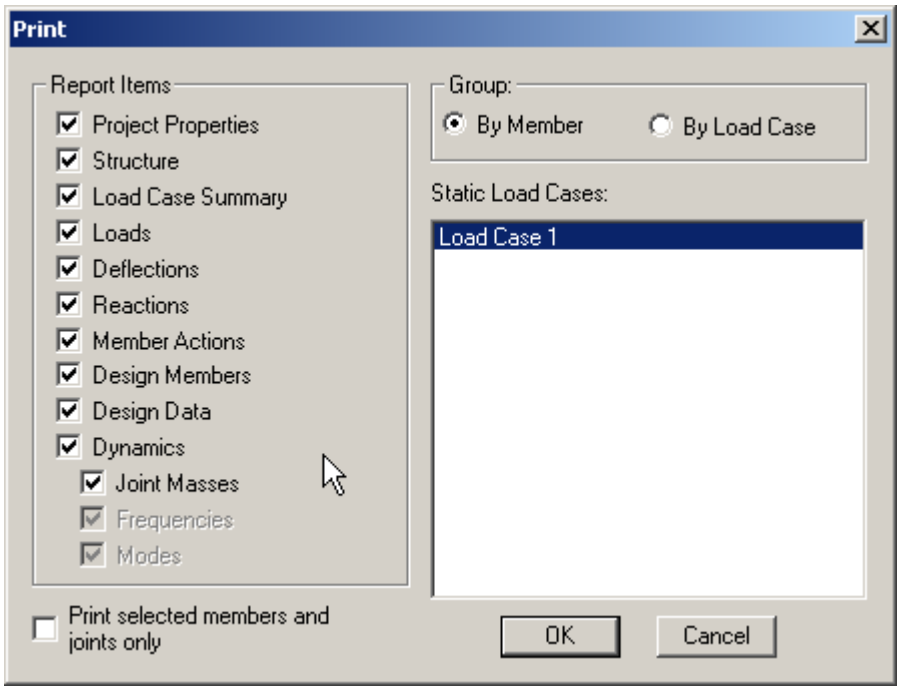
Colour printing is supported on Postscript printers that provide colour output.

Printing Reports

When printing a report summary of Multiframe data, you can print data for the whole frame or restrict the output of results to just the members selected in the front window.

To print a report

- **Optionally, select the members in the front window**
- **Choose Print Summary from the File menu**



- **Turn on the "Print selected members only" check box if you only want output for selected parts of the frame**
- **Click the OK button**

The Print Preview dialog will then appear. You can use the Titles button in this dialog to set optional header and footer information such as file name, date, time etc.

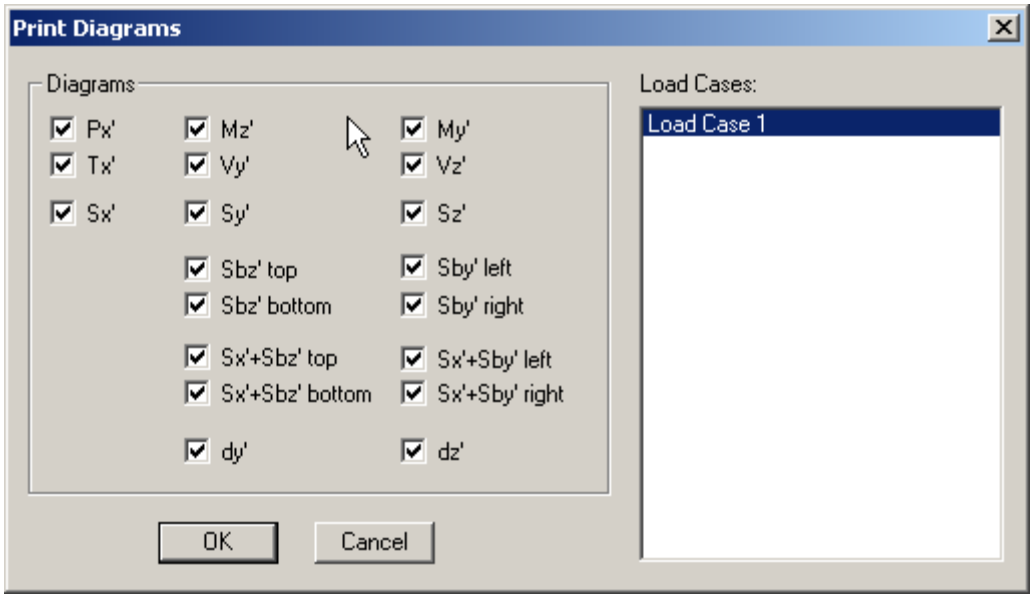
- **Click the Print button to print out the report.**

Printing Diagrams

You can print out a list of member diagrams for the members selected in the front window.

To print member diagrams for a range of members

- **Select the members in the front window**
- **Choose Print Diagrams from the File menu**



- **Choose the actions and load cases that you wish to print**
- **Click the OK button**

The Print Preview dialog will then appear. You can use the Titles button in this dialog to set optional header and footer information such as file name, date, time etc.

- **Click the Print button to print out the report.**

Print Window

To print out the contents of the front window

- **Choose Print Window from the File menu**

The Print Preview dialog will then appear. You can use the Titles button in this dialog to set optional header and footer information such as file name, date, time etc.

- **Click the Print button to print out the contents of the window**

Data Exchange

Multiframe can import and export data to and from other applications using a variety of techniques and data formats. The topology of a frame can be exported to CAD packages via DXF files, images can be pasted into reports via the windows clipboard, data can be copied into spreadsheets or other applications, animations can be saved to file and frame data can be exchanged with other analysis/design or detailing programs via a number of different file format. This chapter describes the various techniques and formats that can be used to exchange information with Multiframe.

File Import

Multiframe can import a frame from numerous file formats. Each file format that can be read by Multiframe is described in the following sections.

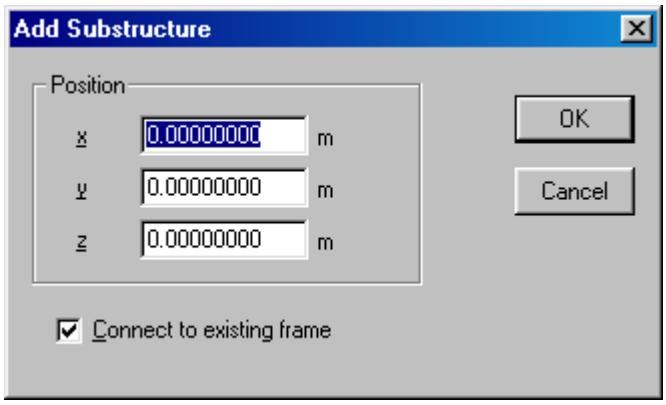
Multiframe Structure

A frame saved as a Multiframe binary file (*.mfd) may be appended to the frame currently open within Multiframe. The

To import a Multiframe file into the existing frame

- **Choose Multiframe Structure… from the Import sub menu**
- **Select the file to be imported using the Open File dialog.**

Upon choosing a file, the Add Substructure dialog will be displayed



- **Type the location at which the origin of the imported frame is to be located.**
- **Choose if the imported frame is to be connected to the existing frame at common nodal points.**
- **Click the OK button**

DXF

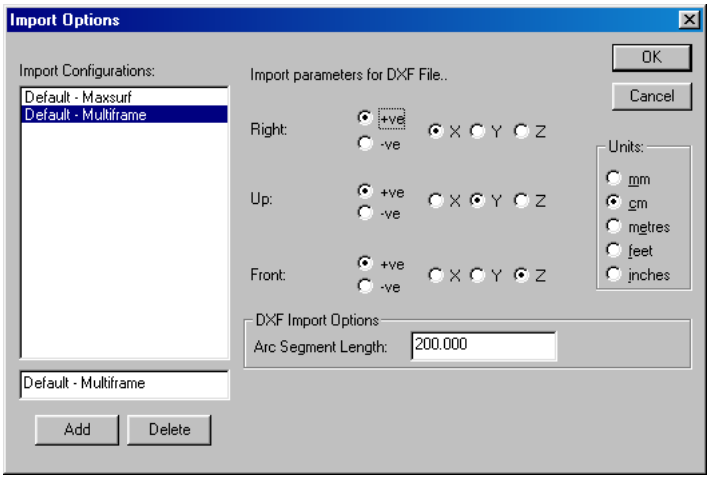
Multiframe can read in 2D and 3D DXF files (Multiframe2D can only read 2D files). This means you can create the geometry for a frame in a CAD program and then save it in DXF format to be read into Multiframe. When reading a DXF file, Multiframe will extract all of the LINE, LINE3D and POLYLINE entities from the file and treat each line segment as a Multiframe member. Lines that are within 0.2 inches (5mm) of each other, will be connected together.
To import a DXF file (*.dxf)

- **Choose DXF… from the Import sub menu**

If a frame is already open within Multiframe you will be asked if you wish to append the DXF file to the existing frame. You may also be prompted to save the existing frame.

- **Select the file to be imported using the Open File dialog.**

Upon choosing a file, the DXF Import Options dialog will be displayed



- **Set the XYZ orientation of the data in the file**

You can use the radio buttons in the middle of the dialog to specify the orientation of the points in the file relative to the Multiframe coordinate system.

- **Set the units of the data in the file**

Click on the radio button corresponding to the units of the data in the file

- **Optionally set the arc segment length**

If you are importing arcs from the file, these will be converted into line segments before importing. This field can be used to specify the length of the line segments along the arc. Note that you can only import ARC objects in the DXF file. Arcs within polylines will not be imported, you should explode the polyline first before importing.

- **Optionally save the configuration**

You can use the Add button at the bottom left of the dialog to add a configuration and save the setting for future usage. For example, if you frequently import from AutoCAD, you might like to set up a configuration named AutoCAD to save your import settings.

- **Click OK to import the data**

When Multiframe imports a DXF file, each line in the file is used to specify a member in the frame. A common problem of importing models from CAD programs is that the each line in the DXF file does not represent a member but rather a number of members. A typical example is a column in a multi-storey building that is often represented as a single line spaning the height of the building. When this occurs it is necessary for the user to subdivide members in the model to ensure intersecting members are connected at a joint. This is readily done in Multiframe using the Intersect Member command.

When importing DXF files, new members imported from the file will be labelled using the drawing layer from which the member originated.

RealDWG

Multiframe can read in 2D and 3D DWG-DXF files. This means you can create the geometry for a frame in a CAD program and then save it in DWG-DXF format to be read into Multiframe. When reading a DWG-DXF file as a background, construction lines or a model, Multiframe will extract all of the LINE, LINE3D and POLYLINE entities from the file and store each line segment as a Multiframe construction line or member. Lines that are within 0.2 inches (5mm) of each other will be connected together.

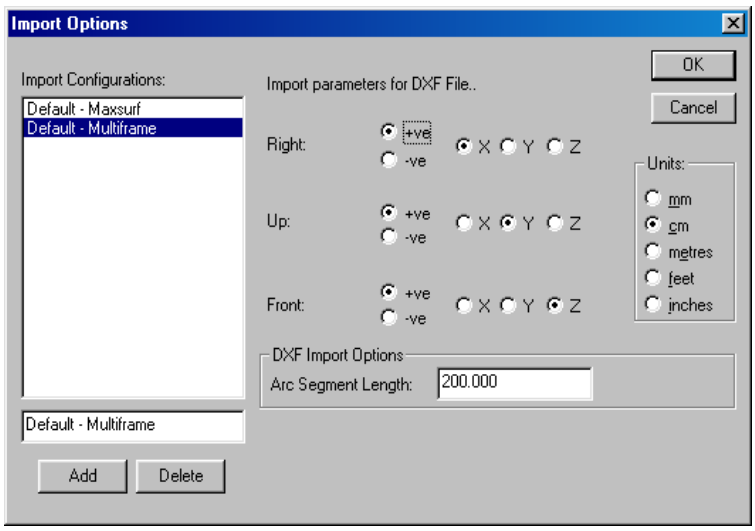
To import a DWG-DXF file (*.dwg or *.dxf)

- **Choose Background DWG-DXF… from the Import sub menu, or**
- **Choose Construct line DWG-DXF… from the Import sub menu, or**
- **Choose Model DWG-DXF… from the Import sub menu**

If a frame is already open within Multiframe you will be asked if you wish to append the DWG-DXF file to the existing frame. You may also be prompted to save the existing frame.

- **Select the file to be imported using the Open File dialog.**

Upon choosing a file, the DWG-DXF Import Options dialog will be displayed



- **Set the XYZ orientation of the data in the file**

You can use the radio buttons in the middle of the dialog to specify the orientation of the points in the file relative to the Multiframe coordinate system.

- **Set the units of the data in the file**

Click on the radio button corresponding to the units of the data in the file

- **Optionally set the arc segment length**

If you are importing arcs from the file, these will be stored as line segments with bulge factors.

- **Optionally save the configuration**

You can use the Add button at the bottom left of the dialog to add a configuration and save the setting for future usage. For example, if you frequently import from AutoCAD, you might like to set up a configuration named AutoCAD to save your import settings.

- **Click OK to import the data**

When Multiframe imports a DWG-DXF file as model, each line in the file is used to specify a member in the frame. A common problem of importing models from CAD programs is that the each line in the DWG-DXF file does not represent a member but rather a number of members. A typical example is a column in a multi-storey building that is often represented as a single line spaning the height of the building. When this occurs it is necessary for the user to subdivide members in the model to ensure intersecting members are connected at a joint. This is readily done in Multiframe using the Intersect Member command.

When importing DWG-DXF files as a model, new members imported from the file will be labelled using the drawing layer from which the member originated.

When Multiframe imports a DWG-DXF file as construction lines, each line in the file is used to store a construction lines. An arc will be store as multi-line segments with bulge factors.

Import Construction Lines

Multiframe can read in 2D and 3D DWG-DXF files as construction lines. This means you can create the geometry for a frame in a CAD program and then save it in DWG-DXF format to be read into Multiframe. When reading a DWG-DXF file as construction lines, Multiframe will extract all of the LINE, LINE3D, ARC and POLYLINE entities from the file and store each line segment as a Multiframe construction line. In order to create patches based on construction lines, it will be automatically broken into 3 pieces of construction lines if a construction line is closed. Multiframe allows users to create members and patches based on construction lines.

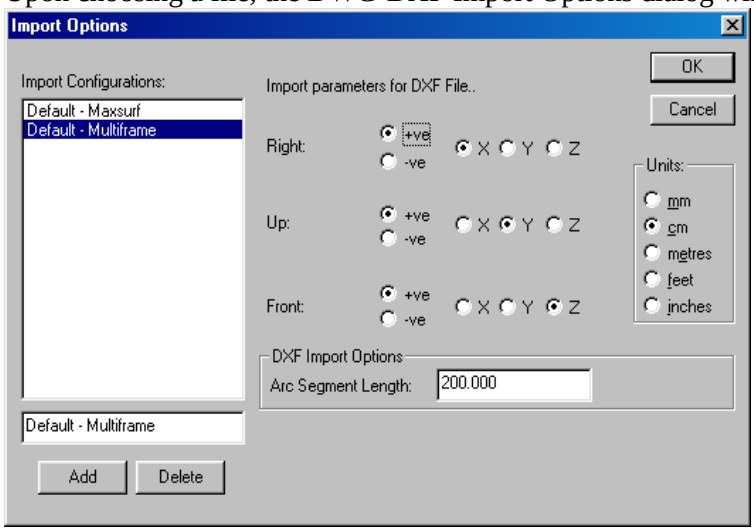
To import a DWG-DXF file (*.dwg or *.dxf) as construction lines

- **Choose Construct line DWG-DXF… from the Import sub menu**

If a frame already has construction lines you will be asked if you wish to delete or append the DWG-DXF file to the existing construction line list.

- **Select the file to be imported using the Open File dialog.**

Upon choosing a file, the DWG-DXF Import Options dialog will be displayed



- **Set the XYZ orientation of the data in the file**

You can use the radio buttons in the middle of the dialog to specify the orientation of the points in the file relative to the Multiframe coordinate system.

- **Set the units of the data in the file**

Click on the radio button corresponding to the units of the data in the file

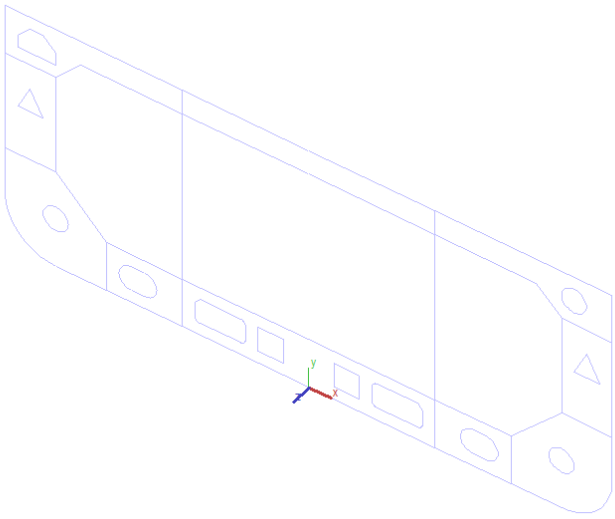
- **Optionally set the arc segment length**

If you are importing arcs from the file, these will be stored as line segments with bulge factors.

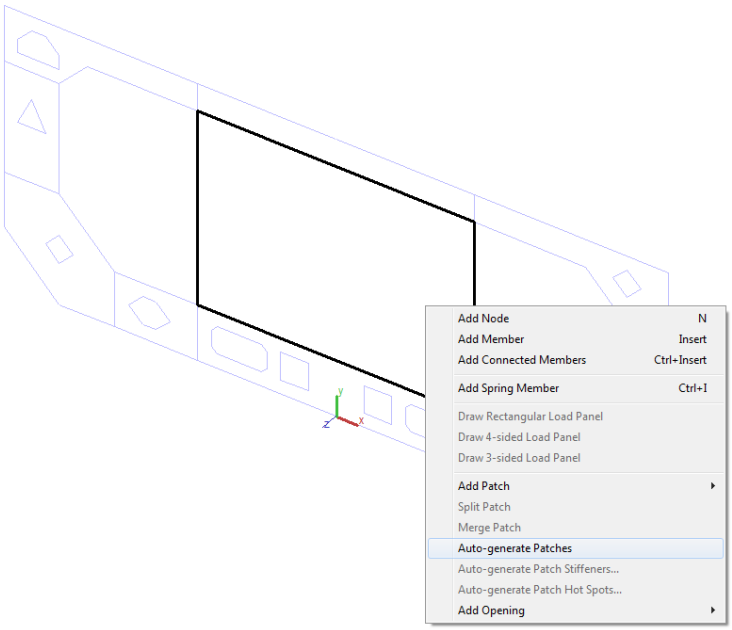
- **Optionally save the configuration**

You can use the Add button at the bottom left of the dialog to add a configuration and save the setting for future usage. For example, if you frequently import from AutoCAD, you might like to set up a configuration named AutoCAD to save your import settings.

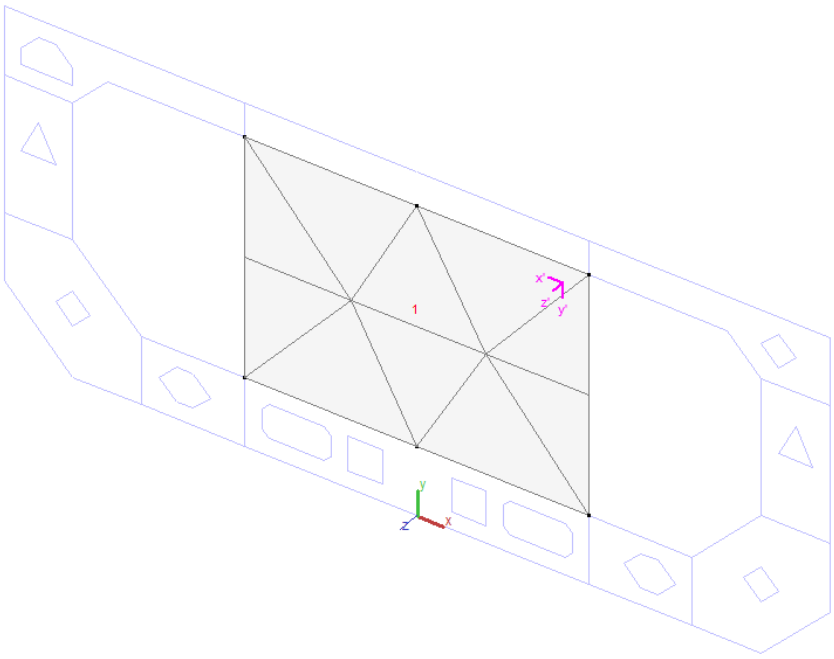
- **Click OK to import the data**



To create patches, select construction lines and mouse right click,



Select ‘Auto-generate Patches’ command and will get:



In a similar way, users can generate members based on construction line selection.

Multiframe Text

This format of Multiframe's text files is described in Appendix D of this document. To import a Multiframe text file (*.txt)

- **Choose Multiframe Text… from the Import sub menu**
- **Select the file to be imported using the Open File dialog.**

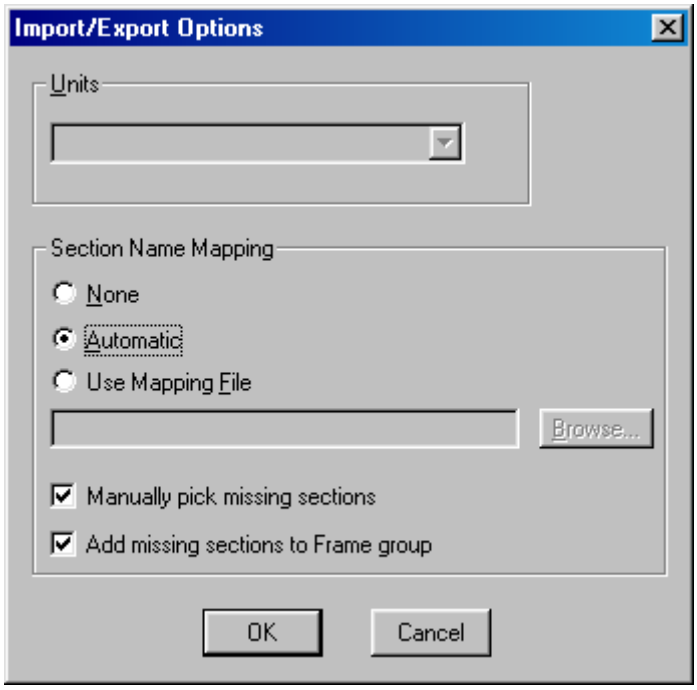
Microstran Archive Files

A Microstran archive file (*.arc) can be opened within Multiframe. Frames imported from Microstran cannot be appended to an existing frame.

To import a Microstran archive file.

- **Choose Microstran Archive … from the Import sub menu**
- **Select the file to be imported using the Open File dialog.**

Upon choosing a file, the Import/Export Options dialog will be displayed



- **Choose how section names in Microstran are to be matched with those currently loaded into Multiframe.**

Section names may be mapped automatically in which case Multiframe will use some simple rules regarding the naming conventions used in Microstran and attempt to modify the names to match a Multiframe section name. Section names may also be mapped using a mapping file that contains a list of Multiframe sections names and an alternative name for that section. Refer to Appendix C for a full description of section mapping files.

- **Choose if the user is to be prompted to manually locate sections within the Microstran file that cannot be resolved to a section with the Multiframe library.**
- **Choose if sections from the Microstran file that are not resolved to a Multiframe section are to be added to the Frame group within the sections library.**
- **Click the OK button**

The Microstran file will now be imported into Multiframe. It is important to note that the data used to describe a frame in Microstran is slightly different in structure to that used in Multiframe. As such some feature and data used within Microstran might be lost when importing into Multiframe. While not all the features in Multiframe and Microstran are compatible, the majority of the geometry, topology and loading can be exchanged via this file format.

The nodes and members in Multiframe are labelled with the original Microstran numbering.

Space Gass Text Files

A Space Gass text file (*.txt) can be opened within Multiframe.

To import a Space Gass text file

- **Choose Space Gass Text … from the Import sub menu**

If a frame is already open within Multiframe you will be asked if you wish to append the file to the existing frame. You may also be prompted to save the existing frame.

- **Select the file to be imported using the Open File dialog.**

Upon choosing a file, the Import/Export Options dialog will be displayed. Complete the settings in this dialog as described above and click OK.

The Space Gass file will now be imported into Multiframe. It is important to note that the data used to describe a frame in Space Gass is slightly different in structure to that used in Multiframe. As such, some feature and data used within Microstran might be lost when importing into Multiframe. While not all the features in Multiframe and Space Gass are compatible, the majority of the geometry, topology and loading can be exchanged via this file format.

The nodes and members in Multiframe are labelled with the original Space Gass numbering.

SDNF

Multiframe can import files in the Steel Detailing Neutral File format (SDNF). This can be used to import models generated in other analysis, design or detailing software into Multiframe.

To import an SDNF file

- **Choose SDNF… from the Import sub menu**

If a frame is already open within Multiframe you will be asked if you wish to append the file to the existing frame. You may also be prompted to save the existing frame.

- **Select the file to be imported using the Open File dialog.**

Upon choosing a file, the Import/Export Options dialog will be displayed. Complete the settings in this dialog as described above and click OK.

The SDNF file will now be imported into Multiframe. While not all the features in the SDNF file are represented in Multiframe ever attempt is made when importing the frame to map the members within the SDNF file to appropriate representations within Multiframe.

Struclink

Multiframe can import files from STAAD and Prosteel using the Struclink (*.struclink) data format

To import a Struclink file

- **Choose Struclink… from the Import sub menu**

Upon choosing a file, Multiframe will automatically detect if it originated from STAAD or Prosteel.
If sections saved in that file are from a country section library different from the Multiframe section library, a prompt will appear asking to continue or not.

Even with the correct section library open it is possible that not all sections will be successfully mapped. If this is the case a warning message will be shown.

File Export

Multiframe can export frames in a number of file formats. Each file format that can be exported by Multiframe is described in the following sections.

DXF 2D/3D

Multiframe can export 2D or 3D DXF files which can be read by AutoCAD and other CAD systems. Depending on what is displayed on screen, the file will contain either lines representing the members in the frame, or polygons representing the shapes of the sections that make up the members. Normally, lines only will be output. However, if you save a DXF file while rendering is turned on in the front window, the more complex polygon option will be used. The polygon option saves the detailed models of the members as 3DFACE entities. This is particularly useful for exporting data to rendering programs that need a polygonal format to do a good job of rendering the frame.

When exporting a DXF files only active members (i.e. not clipped or masked) are included in the exported data. In addition, nodes and members will be labelled with their number and/or label. These will be written to the DXF file if they are visible within Multiframe

To export a frame saved in DXF format

- **Choose 2D DXF… or 3D DXF… from the Export sub menu**
- **Specify the name and location of the file to be exported using the Save File dialog.**

The export of 2D DXF files is provided only for compatibility with older CAD programs which only accept 2D DXF. The members in the frame are saved as LINE entities.

DXF files exported from Multiframe include the colours of the members by using the current colour settings of the Frame window. Furthermore, if the Frame Window has focus and the Member Legend is visible, the members will be exported using drawing layers, each layer representing an item in the legend.

VRML

VRML is a format used to view 3D models in an Internet web browser. You can also embed these models in web pages to provide online display of your projects.

The VRML export option is only available when the frame is rendered in the active window. To export a frame saved in VRML format

- **Choose VRML… from the Export sub menu**
- **Specify the name and location of the file to be exported using the Save File dialog.**

More information on VRML can be found at http://www.web3d.org

Multiframe Text

Multiframe has the capability of saving files in a text format. This facility is designed to allow pre and post processing programs to transfer information to and from Multiframe. The file may also be used as a convenient summary of the data in a human readable format.

To export a frame saved in Multiframe's text file format

- **Choose Multiframe Text… from the Export sub menu**
- **Specify the name and location of the file to be exported using the Save File dialog.**

This format of the Multiframe's text file is described in Appendix D of this document.

Spreadsheet Text

The Spreadsheet text output is a summary of maximum actions for each member in the frame and is in a format suitable for input to a spreadsheet such as Excel or Lotus.

To export a frame saved in the Spreadsheet text file format

- **Choose Spreadsheet Text… from the Export sub menu**
- **Specify the name and location of the file to be exported using the Save File dialog.**
- **Choose which load cases are to be exported**
- **Click OK**

Daystar Text

Multiframe outputs a tabular summary of member actions in a format suitable for input to the Day Star Text AISC steel code checking program available from DayStar Software Inc.

To export a frame saved in the Daystar Text file format

- **Choose Daystar Text… from the Export sub menu**
- **Specify the name and location of the file to be exported using the Save File dialog.**

More information about the Daystar's software can be obtained form their web site at http://www.daystarsoftware.com

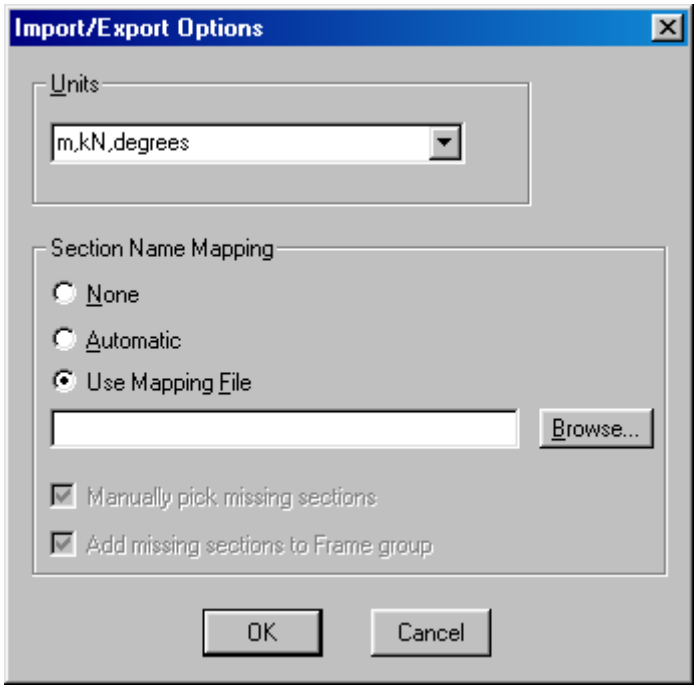
Microstran Archive Files

Multiframe supports exporting the Microstran archive file format (version 4). While not all the features in Multiframe and Microstran are compatible, the majority of the geometry, topology and loading can be exchanged via this file format.

To export a frame saved in the Spreadsheet text file format

- **Choose Microstran Archive… from the Export sub menu**
- **Specify the name and location of the file to be exported using the Save File dialog.**

Upon choosing a file, the Import/Export Options dialog will be displayed



- **Select the units to be used in the exported file.**
- **Choose how section names in Microstran are to be matched with those currently loaded into Multiframe.**

Section names may be mapped automatically in which case Multiframe will use some simple rules regarding the naming conventions used in Microstran and attempt to modify the names of section in Multiframe match the Microstran section names. Section names may also be mapped using a mapping file that contains a list of Multiframe sections names and an alternative name for that section. Refer to Appendix C for a full description of section mapping files.

- **Click the OK button**

The Microstran file will now be exported from Multiframe. It is important to note that the data used to describe a frame in Microstran is slightly different in structure to that used in Multiframe. As such some feature and data used within Multiframe may be lost when exporting to Microstran.

Space Gass Text Files

To export a frame saved in the Spreadsheet text file format

- **Choose Space Gass Text… from the Export sub menu**
- **Specify the name and location of the file to be exported using the Save File dialog.**

Upon choosing a file, the Import/Export Options dialog will be displayed. Complete the option in this dialog as described above and press OK.

The Space Gass file will now be exported from Multiframe. It is important to note that the data used to describe a frame in Space Gass is slightly different in structure to that used in Multiframe. As such some feature and data used within Multiframe may be lost when exporting to Space Gass.

SDNF

Multiframe can export a frame in the Steel Detailing Neutral File format (SDNF). This can be used to import the frame in to other analysis, design or detailing packages.

To export a frame saved in the SDNF format

- **Choose SDNF… from the Export sub menu**
- **Specify the name and location of the file to be exported using the Save File dialog.**

Upon choosing a file, the Import/Export Options dialog will be displayed. Complete the option in this dialog as described above and press OK to export the frame.

Multiframe v7

The release of Multiframe version 7.5 was accompanied by a significant upgrade of the Multiframe binary file format (*.mfd). Files saved in the new format are not compatible with older versions of Multiframe or with the Macintosh version of Multiframe. To provide compatibility with these versions of Multiframe, frames can be saved in the file format used prior to version 7.5.

To export a frame saved in the Multiframe version 7 file format

- **Choose Multiframe v7… from the Export sub menu**
- **Specify the name and location of the file to be exported using the Save File dialog.**

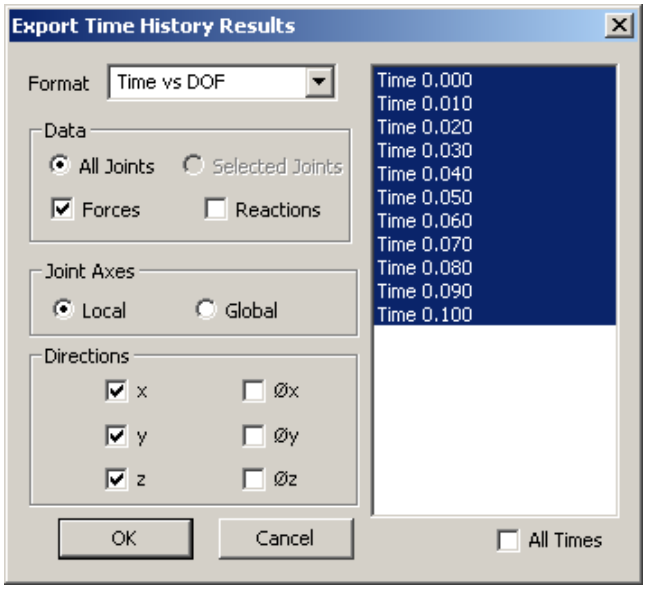
Multiframe will now export the frame in this file format. Some of the data describing the frame may be lost, as this information is not supported in the old file format. The results of analyses are also omitted from the exported file.

Time History Results

The results of a time history analysis can be exported to a text file. To export a frame saved in the Spreadsheet text file format

- **Choose Time History Results… from the Export sub menu**
- **Specify the name and location of the file to be exported using the Save File dialog.**

Upon choosing a file, the Export Time History Results dialog will be displayed. Complete the option in this dialog as described above and press OK.



- **Click OK.**

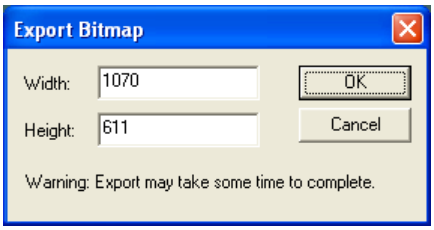
The data to be exported can be controlled by the user who can select which joints, degrees of freedoms and time steps to be saved in the text file. The displacements and/or reactions to be exported can be written to the text file in one of the following 4 formats:

- **Time vs DOF – data is exported as a single table with each row representing a single time step and each column representing a single degree of freedom**
- **Node vs DOF – data is exported with a separate table for each time step, each row in the table contains the data for a single joint.**
- **Nodal Database – data is exported as a single table with each row representing the data for a particular time step for a single joint**
- **DOF data base – data is exported as a single table with each row representing the data for a particular time step for a single DOF**

Bitmap Image……

Bitmap export gives you the ability to export rendered Frame, Plot and Load views to a bitmap file. This is similar to Edit | Copy (Copy to Clipboard) however, the File | Export | Bitmap Image allows you to specify the size of the image to be exported; the larger the size, the better the quality. . To export the image

- **Choose Bitmap Image… from the Export sub menu**



- **Enter the width and height, in pixels, of the bitmap image to be created.**
- **Click OK.**

Note that the aspect ratio of the current view is preserved when you change the width and height.

- **Specify the name and location of the file to be exported using the Save File dialog.**

- **Click OK.**

A bitmap image of the current view will now be saved to the specified file..

Navisworks

NavisWorks provides software for real-time walk-through and inspection of large CAD models. This is particularly useful for presentations and for checking very large models. The model exported not only contains the correct geometry and colours from the Multiframe model, it also contains member attributes such as section name, length etc. More information on NavisWorks can be found at www.navisworks.com. The export of a model in the format requires that a licensed version of either NavisWorks Roamer or NavisWorks Publisher to be installed on your computer.

To export a NavisWorks model,

- **Choose Navisworks from the Export sub menu**
- **In the Save As dialog choose to export either a *.nwc file or a *.nwd file**
- **Click OK.**

A NWC file requires only a NavisWorks Roamer licence and the resulting .nwc files can be read by NavisWorks Roamer on any machine. An NWD file export requires that you have a NavisWorks Publisher licence installed on your computer but the resulting .nwd file can be read by anyone who has the free NavisWorks Freedom viewer. This is a very good way to distribute a viewable model to any other person.

Struclink

It is possible to export Multiframe models to the Struclink format that is compatible with STAAD and Prosteel.

To export to Struclink,

- **Choose Struclink from the Export sub menu**

After selecting a file name a dialog will appear to choose if the export is to Prosteel or to STAAD. After clicking OK the export will start.

ISM

Multiframe can export a frame in Bentley ISM format. This can be used to import the frame in to other Bentley analysis, design or detailing packages such as Staad.Pro, Staad(x) & Sacs etc.

To export a frame saved in the ISM format

- **Choose ISM from the Export sub menu**
- **Specify the name and location of the file to be exported using the Save File dialog.**

Graphics

The image in any of Multiframe graphical windows can be copied to the clipboard using the Copy command in the Edit menu or by pressing Ctrl-C. This provides a simple means for using pictures of your model or plots of member actions into other applications.

Table Data

Tables of numbers can be copied directly from the Data and Results windows onto the clipboard using the Copy command from the Edit menu or via the Ctrl-C key combination. If you hold down the Shift key in the Data or Result window while choosing the Copy command from the Edit menu, the column titles will be included in the text placed on the clipboard. This information is in a format that allows it to be pasted directly into a spreadsheet or used for constructing a table in a word processor. A discussion on how a spreadsheet can be using in conjunction with Multiframe is described in Appendix E.

You can paste tables of numbers into Multiframe in the data window by preparing a table of data in a spreadsheet and then pasting it into the selected cells in the Data window. When pasting in numbers, make sure that the numbers to be pasted have the same number of rows and columns as the selected area in the Data window.

Animations

Multiframe provides a capability for creating a movie from any animation that can then be inserted into any AVI-aware program. This is useful for keeping a record of an analysis or for linking up with a post-processing program that could be used for visualization. To save a movie, turn on the Save Animation File option in the Animate dialog. Multiframe will create and play the animation as usual, but will also save a movie which can then be placed or played in any AVI aware application.

Chapter 3 Multiframe Reference

This chapter summarises the overall structure, windows, toolbars and menu commands of Multiframe.

- Windows
- Toolbars
- Menus

Windows

Multiframe uses a range of graphical, tabular, graph and report windows.

Frame Window

This window is used for preparing a physical description of the structure. This includes geometry, connections, section types, section orientation and restraints. You can also modify joint and member masses for dynamic analysis.

Data Window

This window is used for viewing and editing the data describing the structure and its loading. It allows you to edit data numerically rather than graphically. A number of different tables of data can be displayed describing joints, member geometry, member properties, joint loads, member loads, restraints and prescribed displacements, springs, sections and load cases.

Load Window

This window is used for setting up the loading conditions you wish to apply to the structure. You can also add self weight load cases and control whether member's self weight is included in static analysis.

Result Window

This window is used for viewing the results of the analysis in numerical form. It can display tables of joint displacements and reactions and member actions or end forces. Results of any modal analysis are also displayed here in the form of a table of frequencies and periods of vibration.

Plot Window

This window is used for viewing diagrams of the forces and deflections in the structure. It allows you to view diagrams of the whole structure, individual members, joint reactions and joint displacements. You also view the mode shapes for modal analysis in this window.

CalcSheet Window

This window is used for preparing and evaluating design calculations that use the results of the analysis as a basis for design checks.

Report Window

This window is used for viewing analysis details or design calculations when Steel Design is also installed.

Toolbars

You can use the icons on the toolbars to speed up access to some commonly used functions. You can hold your mouse over an icon to reveal a pop-up tip of what the icon does.

File Toolbar

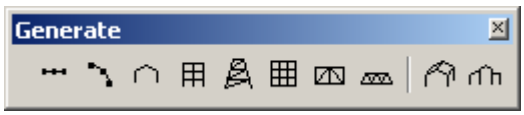
The File Toolbar provides shortcuts to a number of commands frequently associated with Windows applications. From left to right, the buttons generate the following geometry:

New
Open
Save

Cut
Copy
Paste

Print
Help

Generate Toolbar

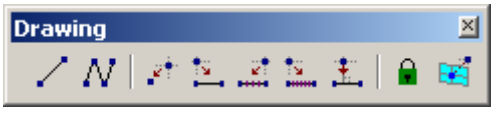


The Generate Toolbar is found in the Frame Window and performs the same functions as the Generate command in the Geometry menu. From left to right, the buttons generate the following geometry:

Continuous Beam
Curved Member
2D Portal Frame
Multi-story Frame
Trestle
Grillage
Pratt Truss
Warren Truss

3D Portal
Multi-bay building

Drawing Toolbar



The Drawing Toolbar is found in the Frame Window and provides shortcuts to commands for adding members to the model and associated settings. From left to right, the buttons perform the following functions:

Add a Member
Add a continuous Member

Toggle snap to joints
Toggle snap to members
Toggle snap to member quarter points
Toggle snap to user-defined increments on member
Toggle perpendicular snap to member

Lock topology of model
Constrain drawing to current drawing depth

Geometry Toolbar

The Geometry Toolbar is found in the Frame Window and provides commands to manipulate the geometry of the frame graphically. From left to right, the buttons perform the following functions:

Move
Rotate
Mirror
Shear
Shear

Extrude
Duplicate
Rescale

Member Toolbar

The Member Toolbar is found in the Frame Window and performs some of the functions found in the Frame and Geometry menus. From left to right, the buttons perform the following functions:

Delete Member
Sub-divide Member

Select section type from most recently used sections
Section Type dialog

Pinned/Pinned Member Release
Rigid/Rigid Member Release

Member Orientation

Joint Toolbar

The Joint Toolbar is found in the Frame Window and performs some of the functions found in the Frame menu. From left to right, the buttons perform the following functions:

Fixed Joint Restraint
Pinned Joint Restraint
Horizontal Roller Restraint
Vertical Roller Restraint
Free Joint Restraint

Rigid Joint Type
Pinned Joint Type

View Toolbar

The View Toolbar is found in the Frame, Load, and Plot Windows and performs some of the functions found in the View menu. From left to right, the buttons perform the following functions:

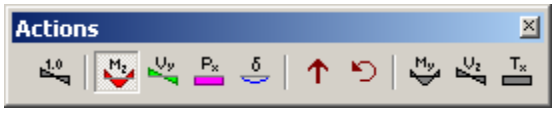
Select rectangular selection tool
Select linear selection tool

Zoom
Shrink
Pan
Size To Fit

Toggle Clipping
Toggle Masking

Toggle Drawing Grid
Toggle Structural Grid
Toggle Axes

Actions Toolbar



The Actions Toolbar is found in the Plot Window and performs some of the functions found in the Display menu. From left to right, the buttons perform the following functions:

Display plot values

Bending Moment about the local z' axis
Shear forces in the local y' axis
Axial forces for the current load case
Display Deflection

Display force reactions
Display moment reactions

Bending Moment about the local y' axis
Shear forces in the local z' axis
Torque about the local x' axis

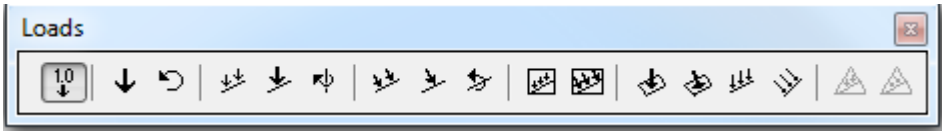
Load Case Toolbar

The Load Case toolbar can be used as a shortcut for changing the current load case. You can select a load case from the drop down list or click the right and left arrows to go to the next or previous load case.

The first button on the toolbar provides access to a Re-analyse command which analyses the model using the settings last input via the Analyse dialog.

The button on the far right of the toolbar is used to switch between the linear and nonlinear results for the current load case.

Load Toolbar



The Load Toolbar is found in the Load Window and provides shortcuts to command associated with loads. From left to right, the buttons perform the following functions:

Display load labels

Joint Load
Joint Moment

Global member distributed load
Global member point load
Global member point moment

Local member distributed load

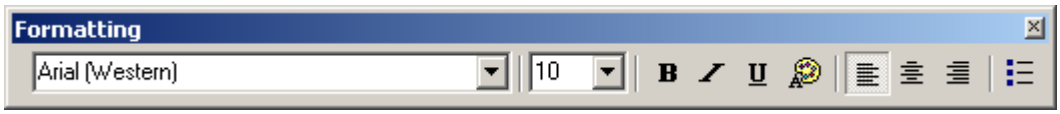
Local member point load
Local member point moment

Global load panel distributed load
Local load panel distributed load

Global patch point load
Local patch point load
Global patch distributed load
Local patch distributed load

Global plate distributed load
Local plate distributed load

Formatting Toolbar



The Formatting Toolbar is used to change the attributes of the text displayed in each of the windows in Multiframe. It also provides shortcuts to commands for formatting text within the Report Window. From left to right, the buttons perform the following functions:

Font Name
Font Size
Bold
Italic
Underlined
Colour

Align Left
Align Centre
Align Right

Bullets

Symbols Toolbar

The Symbols Toolbar is available when the Frame, Load, or Plot Window is the front window. It toggles the display of some of the joint, member and section symbols found in the Symbols dialog. This toolbar also contains a button to toggle rendering of the structure. From left to right, the buttons show or hide the following symbols:

Joint numbers
Joint labels
Joint restraints

Member numbers
Member labels

Member axes
Member releases

Section names
Section axes

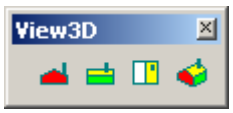
Legends

Windows Toolbar

The Windows Toolbar can be used to quickly switch from one window to another. From left to right, the buttons show the following windows:

Frame, Data, Load, Result, Plot, CalcSheet, Report.

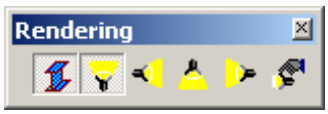
View3D Toolbar



The View3D Toolbar can be used to quickly switch from one view to another while you are working in any of the graphics windows. It lets you switch to the 4 most commonly used views. You can also use the View button at the bottom left of the window to switch to more views. From left to right, the buttons show the following views:

Front, Right, Top, 3D.

Render Toolbar



The Rendering Toolbar can be used to quickly turn rendering on and off. It also provides a shortcut to editing the lighting setup in OpenGL. The four lights that may be used for OpenGL rendering may be toggled on or off or the lighting properties edited.

Toggle Rendering, Light 1 On/Off, Light 2 On/Off, Light 3 On/Off, Light 4 On/Off, Customise Lights

Group Toolbar

The Groups toolbar can be used as a shortcut for changing the current group set and allows you to select a group set from the drop down list.

The button on the far right of the toolbar is used to Add a group to the current group set.

Clipping Toolbar

The Clipping Toolbar available for the Graphic views, it provides shortcuts to frequently used clipping commands. From left to right, the buttons perform the following functions:

Toggle Clipping on/off
Clip to Zone
Clip to Frame

Load Panel Toolbar

The Load Panel Toolbar available for adding panels and rotating support edges. It provides shortcuts to frequently used load panel commands. From left to right, the buttons perform the following functions:

Add rectangle load panel
Add 4-node load panel
Add 3-node load panel
Auto-generate load panels
Rotate panel supports

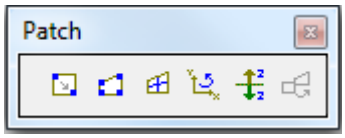
Load Panel Symbols Toolbar

The Panel Symbols Toolbar is available when the Frame, Load, or Plot Window is the front window. It toggles the display of some of the load panels found in the Symbols dialog. From left to right, the buttons show or hide the following symbols:

Panel axes
Panel numbers
Panel labels
Panel support edges
Panel load tributary areas

Panel loads
Panel corner/edge loads (preview)

Patch Toolbar



The Patch Toolbar available for adding patches, rotating local axes, flipping z-axis and converting load panel to patches. It provides shortcuts to frequently used patch commands. From left to right, the buttons perform the following functions:

Add rectangle patch
Add N-node patch
Auto-generate patches
Rotate patch local axes
Flip patch z-axis
Convert load panels to patches

Patch Symbols Toolbar

The Patch Symbols Toolbar is available when the Frame, Load, or Plot Window is the front window. It toggles the display of some of the patches found in the Symbols dialog. From left to right, the buttons show or hide the following symbols:

Patch visible
Patch internal meshes
Patch edge nodes
Patch internal meshed nodes

Patch colour fill
Patch edge lines
Patch axes
Patch numbers
Patch labels
Patch materials
Patch thickness
Patch loads

Plate axes
Plate numbers
Plate labels
Plate materials
Plate thickness
Plate loads

Menus

Multiframe uses the standard set of Windows menu commands for File, Edit and Windows operations. It also has a range of menus for other commands.

- File Menu
- View Menu
- Select Menu
- Geometry Menu
- Group Menu
- Frame Menu
- Display Menu
- Patch Internal Forces Submenu

The items in the Patch Internal Forces submenu may be used to control which type of internal force is displayed in the Plot window.

Moment Mxx'

Display the computed plate bending moments about the local xx' axis for the current load case in the Plot window.

Moment Myy'

Display the computed plate bending moments about the local yy' axis for the current load case in the Plot window.

Moment Mxy'

Display the computed plate twisting moments in x-y' plane for the current load case in the Plot window.

Shear Vxz'

Display the computed plate transverse shear force about the local z' axis in x-z' plane for the current load case in the Plot window.

Shear Vyz'

Display the computed plate transverse shear force about the local z' axis in y-z' plane for the current load case in the Plot window.

Moment Mmin

Display the computed minimum plate bending moments about the principal axis for the current load case in the Plot window.

Moment Mmax

Display the computed maximum plate bending moments about the principal axis for the current load case in the Plot window.

Force Fxx'

Display the computed membrane direct forces about the local xx' axis for the current load case in the Plot window.

Force Fyy'

Display the computed membrane direct forces about the local xx' axis for the current load case in the Plot window.

Force Fxy'

Display the computed membrane shear forces in x-y' plane axis for the current load case in the Plot window.

Force Fmin

Display the computed minimum membrane forces about the principal axis for the current load case in the Plot window.

Force Fmax

Display the computed maximum membrane forces about the principal axis for the current load case in the Plot window.

Patch Stresses Submenu

The items in the Patch Stresses submenu may be used to control which type of patch stress is displayed in the Plot window.

Sxx'

Display the computed in-plane direct stresses about the local xx' axis for the current load case in the Plot window.

Syy'

Display the computed in-plane direct stresses about the local yy' axis for the current load case in the Plot window.

Szz'

Display the computed transverse direct stresses about the local zz' axis for the current load case in the Plot window.

Sxy'

Display the computed in-plane shear stresses in x-y' plane for the current load case in the Plot window.

Sxz'

Display the computed transverse shear stresses in x-z' plane for the current load case in the Plot window.

Syz'

Display the computed transverse shear stresses in yz' plane for the current load case in the Plot window.

Von Mises

Display the computed Von Mises stresses for the current load case in the Plot window.

Tresca

Display the computed Tresca stresses for the current load case in the Plot window.

- **Case Menu**
- **Window Menu**
- **Help Menu**

File Menu

The File menu contains commands for opening and saving files, importing and exporting data, and printing.

New

(Ctrl N)
Erases the structure and all loads. Use this to start work on a new structure. If you have any work unsaved, Multiframe will prompt you to save any changes to the structure or calculations before starting the new work.

Open

(Ctrl O)
Open a file that has previously been saved on disk. If the CalcSheet window is in front, it will read in a new calculations file. Otherwise, it will read in a frame you have previously saved to disk.

If you have any work unsaved, Multiframe will prompt you to save any changes to the structure or calculations before opening the new file. Multiframe can read Multiframe, Multiframe text and DXF files.

Close

Select Close when you wish to finish with the current design. Before closing, a dialog box will appear asking whether you wish to save the current design. If you select Yes the current design will be saved to the disk.

Save

(Ctrl S)
Save work so far with the same name you saved with last time.

Save As

Save work so far in a file with a new name. If the CalcSheet window is in front, this will save the current calculations in a file. Otherwise, it will save the structure.

Import

See "Import Submenu"

Export

See "Export Submenu"

Open Library (Multiframe4D)

The Open Library command provides a submenu which allows the user to open a Sections Library or Load Library.

Open Sections Library (Multiframe2D and 3D)

This command opens a Sections Library.

Page Setup

Set up the printer for printing.

Print Summary

Print out the data for the frame and results of the analysis.

Print Diagrams

Print out member diagrams for the selected members in the Plot window.

Print Window

Print the contents of the front window on the screen

Properties

The Properties command allows you to save project and designer information with your file. This information will also appear at the top of your printed reports.

The Properties dialog also contains two additional pages that display properties of the file and statistics about the frame.

Exit

Leave Multiframe and go back to the Desktop. If you have any work unsaved, Multiframe will prompt you to save any changes to the structure or calculations before quitting.

Import Submenu

Import data from another program into Multiframe.

Multiframe Structure

Imports the structure contained in an existing Multiframe file into the current frame. Loads are not imported.

DXF

Files can be exported in either 2D or 3D format. This is to allow compatibility with older CAD programs that only accept 2D DXF. The members in the frame are saved as LINE entities.

Multiframe will allow DXF file input from any CAD program. We have tested with AutoCAD, MiniCAD, Microstation and Claris CAD, and we expect input from all other systems to work also.

When reading in DXF files, Multiframe will interpret each line or segment of a polyline as a member. Any members that have ends within 0.2 in (5mm) of each other, will be connected together. Multiframe can read 2D or 3D DXF files compatible with AutoCAD release 10 and higher.

Multiframe Text

Imports a file in Multiframe text file format. See “Appendix D”.

Microstran Archive

Multiframe supports importing the Microstran archive file format (version 4). While not all the features in Multiframe and Microstran are compatible, the majority of the geometry, topology and loading can be exchanged via this file format.

Space Gass Text

Multiframe supports importing the Space Gass text file format. While not all the features in Multiframe and Space Gass are compatible, the majority of the geometry, topology and loading can be exchanged via this file format.

SDNF Text

This command will import a file written using the Steel Detailing Neutral File format (SDNF).

Export Submenu

Data can be exported from Multiframe in file formats compatible with other programs.

2D DXF

This is to allow compatibility with older CAD programs which only accept 2D DXF. The members in the frame are saved as LINE entities. Otherwise all data is the same as is described in the next section.

3D DXF

If you do not have rendering turned on in the front window when you save in 3D DXF format, the lines representing the frame will be saved as 3DLINE entities.

If rendering is turned on, 3DFACE entities will be saved in the file. This is useful for input to rendering or 3D presentation or modelling programs.

When exporting, only active members (i.e. not clipped or masked) are included in the exported data. In addition, node and member labels will be written to the DXF file if they are visible within Multiframe.

When the Member Legend is visible the DXF export will utilise a separate layer for each item in the legend. Members associated with each legend item will exported within to corresponding layer.

VRML

The VRML export option is only available when the frame is rendered in the active window. VRML is a format used to view 3D models in an Internet web browser. You can also embed these models in web pages to provide online display of your projects. More information on VRML can be found at http://www.web3d.org

Multiframe Text

Exports a file in Multiframe text file format. See “Appendix D”.

Spreadsheet Text

There are two types of text output. Multiframe Text described above is best for input to a post-processing program. The Spreadsheet text output is a summary of maximum actions for each member in the frame and is in a format suitable for input to a spreadsheet such as Excel or Lotus.

Day Star Text

Multiframe outputs a tabular summary of member actions in a format suitable for input to the Day Star Text AISC steel code checking program available from DayStar Software Inc., http://www.daystarsoftware.com

The American steel library included with Multiframe uses a section naming convention consistent with that used by DS Steel.

Microstran Archive

Multiframe supports exporting the Microstran archive file format (version 4). While not all the features in Multiframe and Microstran are compatible, the majority of the geometry, topology and loading can be exchanged via this file format.

Space Gass Text

Multiframe supports exporting the Space Gass text file format. While not all the features in Multiframe and Space Gass are compatible, the majority of the geometry, topology and loading can be exchanged via this file format.

SDNF Text

This will export a file in the Steel Detailing Neutral File format (SDNF).

Multiframe v7

This will export your Multiframe file in a previous file format which can be read into Multiframe for Windows versions prior to version 7.5 and Multiframe for Macintosh version 4.0 and later.

Time History Results

This will export the results of the current Time History load case to a text file.

Bitmap Image

This will export a bitmap image of the active graphical window. This option is only available when OpenGL rendering is active.

NavisWorks

Exports an OpenGL rendering of the model in one of the NavisWorks file formats for use in real-time walk-throughs and inspections of large CAD models. The export of a model in one of these formats requires that a licensed version of either NavisWorks Roamer or NavisWorks Publisher to be installed on your computer.

Edit Menu

The Edit menu contains commands for copying and pasting data and working in tables.

Undo

(Ctrl Z)
Undo the last action you carried out. The name of this item will change to reflect the command that can be undone.
Windows users: If your frame as less than 500 members you can undo up to 10 steps.

Redo

(Ctrl Y)
Return to the last action you carried out before selecting Undo from the Edit menu.

Cut

(Ctrl X)
Remove the current selection and place it on the clipboard.

Copy

(Ctrl C)
In the Data, Result or Graphics window, this command copies the current selection to the Clipboard

If you hold down the Shift key in the Data or Result window while choosing the Copy command from the Edit menu, the column titles will be included in the text placed on the clipboard.

If you hold down the Shift key in a graphics window while choosing the Copy command from the Edit menu, you will be presented with a dialog offering options to copy to a Pict, DXF, Renderman, or Postscript File to a specified scale.

Paste

(Ctrl V)
Paste the contents of the clipboard into the current selection.

Clear

Remove the current selection without placing it on the clipboard.

Sections

See "Sections Submenu"

Materials

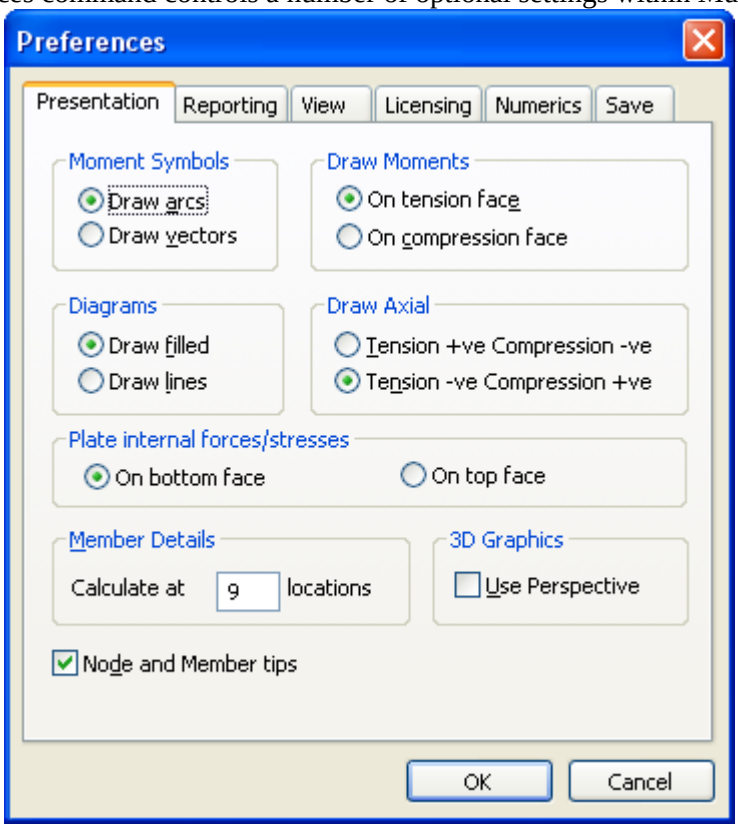
See "Materials Submenu"

Edit Load Library

(Multiframe 4D only)
Allows you to view, edit and add data in the load library.

Preferences

The Preferences command controls a number of optional settings within Multiframe.



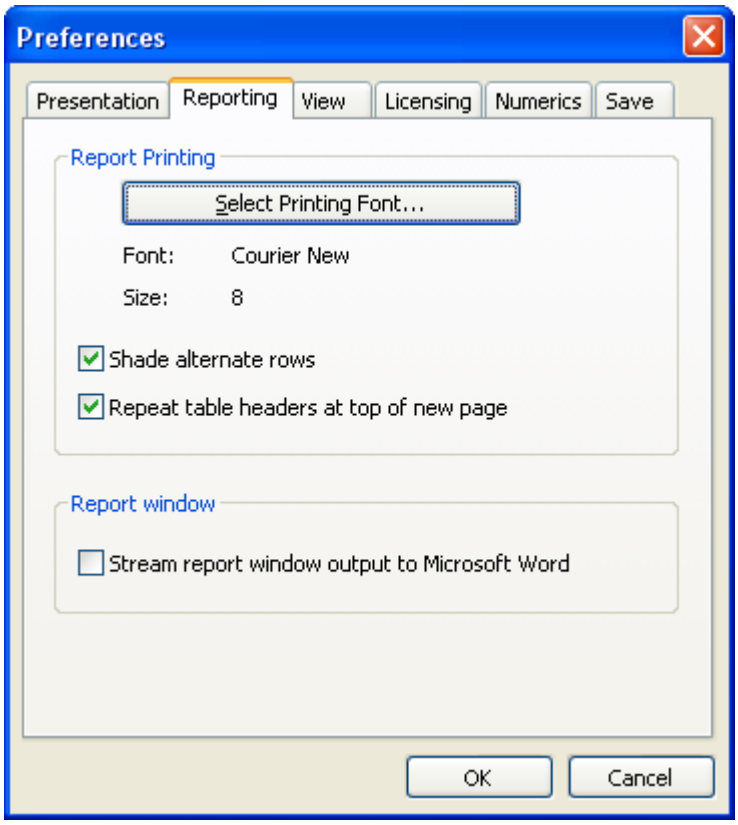
Presentation

The Presentation tab allows you to set how moment diagrams are drawn and what type of moment symbols to use. You can also set the sign convention for axial forces and stresses. You can choose to either display tension or compression as positive when drawing member diagrams.

Stresses are also displayed according to the sign convention you choose. This ensures that the signs of stresses caused by bending correspond with the signs of stresses caused by axial actions.

There is also an option to use perspective when displaying graphics in the 3D views. You may find this useful for presentation purposes. However, you will usually find it more convenient to work with the standard orthographic projection. If you do turn on the perspective option, it uses a perspective angle of 30° corresponding to the natural perspective of the human eye.

Reporting



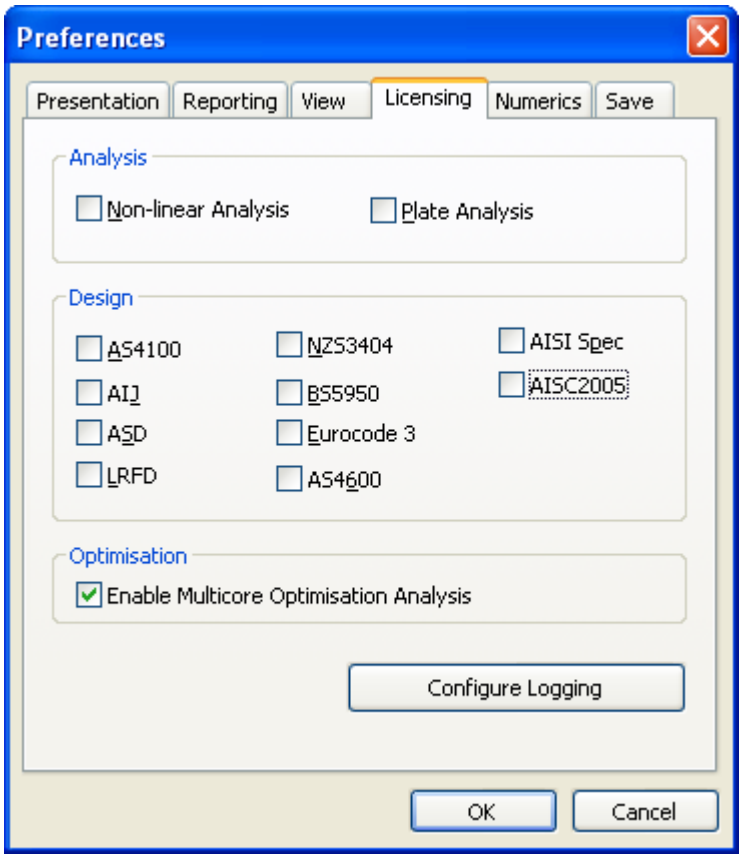
You can control the font and size of text used when printing by using the Reporting tab in the Preferences dialog. The appearance of the report can also be modified to lightly shade every 2nd row in the report which assists is making the tables easier to read. An option is also available to force the header rows of tables within the report to be repeated at the top of a page when the table extends onto a new page.

The reporting in Steel Designer can be redirected to a Microsoft Word document. This option requires that a version of Microsoft Word be installed on the computer. Multiframe will automatically run Microsoft Word when it is required.

View

The View tab contains an option for automatically synchronising selections between the different graphical views.

Licensing

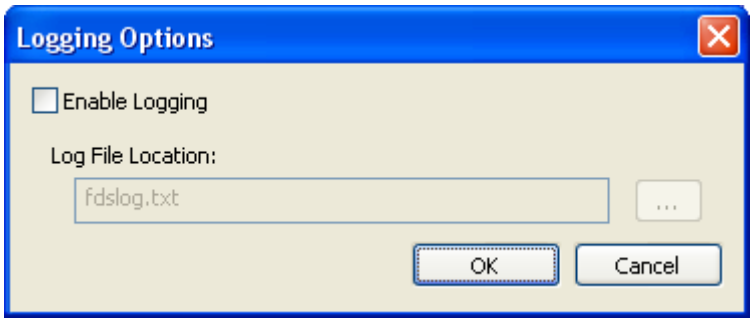


The licensing tab enables or disables the integrated steel design module. To activate Steel Designer the user must own the appropriate licenses and have the correct access codes. Multiframe will need to be restarted for the changes to take effect. You can also control activation of the nonlinear analysis option is a similar way.

Multicore Optimisation Analysis

Multicore Optimistaion will be enabled by default on systems that have multiple cores. When enabled the Multicore option uses all available cpu cores to perform an analysis. This will reduce the time taken to perform an analysis, especially on models containing large numbers of plates.

Logging



Logging is a debug function that allows the recording of user access. When logging has been enabled user usage, license and dongle number and application will be recorded in the specified text file.

Numerics

The Numerics tab is used to set the default format used to display dimensionless numbers in Multiframe. You can choose to use decimal, scientific or engineering notation and specify how many digits of precision you wish to display.

Save

This Save tab allows the option to save a backup file to be turned on or off. By default, the backup is on.

Properties

If the Frame window is foremost, this command shows details about the joint, element or panel selected. If the Load window is foremost, this command shows details about the loads on the selected joint, member, load panels, patches or plates.

Sections Submenu

The Sections menu has commands for adding, editing and deleting sections within the current sections library.

Add Section

Add a new section to the sections library. Sections shapes that are not standard shapes within Multiframe are added using this command and requires the user to input the properties if the section.

Add Standard Section

Add a new section to the sections library. Standard section shapes supported within Multiframe are added by specifying the dimensions of the shape. All section properties will be computed.

Edit Section

Edit the data for a section in the sections library.

Delete Section

Delete a section from the sections library.

Section Colours

Edit the colour associated with each section.

Materials Submenu

The materials menu has commands for adding, editing and deleting materials within the current sections library.

Add Material

Add a new material to the sections library.

Edit Material

Edit the data for material in the sections library.

Delete Material

Delete a material from the sections library.

View Menu

The View menu contains commands for controlling the appearance of the display in the graphical window.

Zoom

(Ctrl W or mouse wheel)
Zoom in on part of the current display. A cross-hair will appear and the view to be viewed in close-up may be selected by pressing the mouse button and dragging a rectangle surrounding the area of interest. Release the button to draw the zoomed view.

Pan

(Ctrl E)
Pan across the structure displayed in the front window. Press and drag in the window to move the frame.

Shrink

(Ctrl R or mouse wheel)
Reduce the size of the drawing in the front window to half its current size.

Size To Fit

(Ctrl T or Home)
Scale the drawing in the front window so that it just fits inside the window. The Size To Fit menu is a sub menu of the View menu, and is described in detail following this section on the View menu.

Clipping

See "Size To Fit Submenu".

Masking

See "Masking Submenu".

Current View

See "Current View Submenu".

Drawing Depth

Set the depth that drawing will occur in the two dimensional views (not available in 2D).

Size

Set the maximum and minimum coordinates available in the Frame window. Use this to set up the overall coordinates before you begin creating a structure.

Grid

Sets the properties of the geometric grid that is projected onto the current drawing plane in the Frame Window to constrain drawing and dragging.

Structural Grids

Opens the Structural Grid Manager dialog that controls the definition, display and interaction of 3 dimensional grids in the Frame Window.

Axes

Turn on or off the display of axes in the front window.

Font

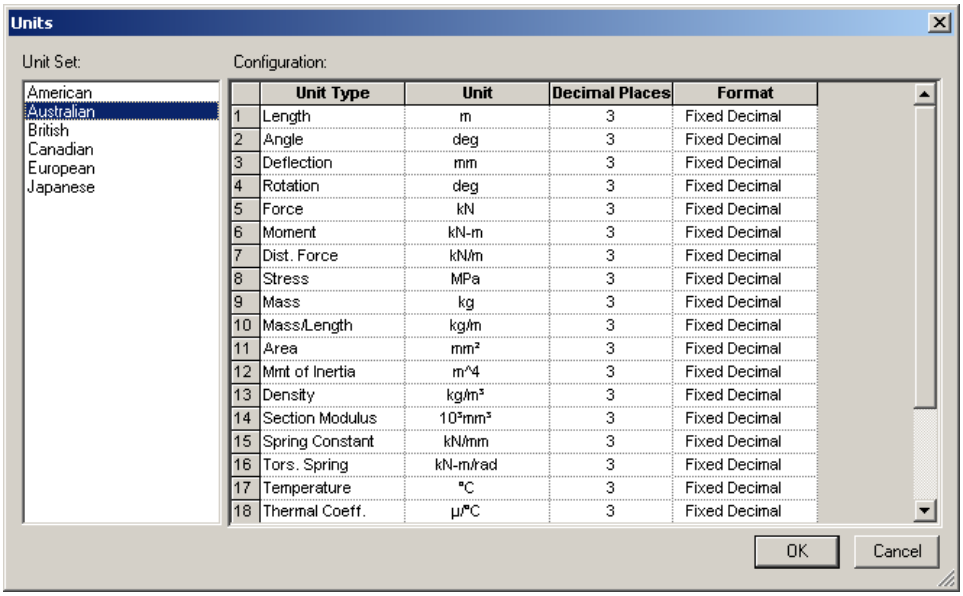
Set the font size and style for the text in the front window.

Units

Multiframe allows you to work in a range of units. Related sets of units are conveniently grouped together under a country name. You can switch from one country setting to another and/or change the units used for any particular item.

To change the units used for display

- **Choose Units… from the View menu**



	Unit Type	Unit	Decimal Places	Format
1	Length	m	3	Fixed Decimal
2	Angle	deg	3	Fixed Decimal
3	Deflection	mm	3	Fixed Decimal
4	Rotation	deg	3	Fixed Decimal
5	Force	kN	3	Fixed Decimal
6	Moment	kN-m	3	Fixed Decimal
7	Dist. Force	kN/m	3	Fixed Decimal
8	Stress	MPa	3	Fixed Decimal
9	Mass	kg	3	Fixed Decimal
10	Mass/Length	kg/m	3	Fixed Decimal
11	Area	mm²	3	Fixed Decimal
12	Mmt of Inertia	m^4	3	Fixed Decimal
13	Density	kg/m³	3	Fixed Decimal
14	Section Modulus	10³mm³	3	Fixed Decimal
15	Spring Constant	kN/mm	3	Fixed Decimal
16	Tors. Spring	kN-m/rad	3	Fixed Decimal
17	Temperature	°C	3	Fixed Decimal
18	Thermal Coeff.	µ/°C	3	Fixed Decimal

- **Click on the name of the set of units you wish to use or change**
- **Use the pop-up menus beside each type of unit to set its units type**
- **Specify the numeric format to be used for each unit type by specifying the number of decimal places and the number format.**
- **Click the OK button**

The units you choose will be saved with the Multiframe application for subsequent use.

Colour

Specify which colours to use drawing the various parts of the structure, for the background of the windows, for the clipped or masked members and for the rendering of the structure.

Status Bar

Makes the Status Bar visible or invisible.

Toolbar

See "Toolbar Submenu"

Size To Fit Submenu

The Size To Fit submenu commands allow you to change the view factor to fit the frame to the current window.

Frame

(Ctrl+Insert)
Resizes the view to fit the entire frame to the window.

Selection

Fit the selection to the window.

Clipping

Fit the active clipping region to the window.

Clipping Submenu

The Clipping submenu commands allow you to control the display and positioning of the clipping bars. Clipping allows you to define how much of the structure is visible at one time.

No Clipping

Turns off clipping if it is on. This hides the clipping bars and makes all members in the frame visible.

Clip Grey

Turns on clipping and makes the clipping mode grey. This means all members which do not lie completely within the boundaries of the clipping bars, will be drawn in grey.

Clip Invisible

Turns on clipping and makes the clipping mode invisible. This means all members that do not lie completely within the boundaries of the clipping bars, will be made invisible.

Clip To Frame

Turns on clipping if it wasn't already on and positions the clipping bars just outside the outermost limits of the frame in each direction. This means all members will be visible.

Clip To Window

Turns on clipping if it wasn't already on and positions the four clipping bars which are visible in the current view so that they lie just inside the boundaries of the window.

Clip To Selection

Turns on clipping if it wasn't already on and positions the clipping bars so that they lie just outside the maximum extents of the selected members in the window.

Clip To Group

Turns on clipping if it wasn't already on and positions the clipping bars so that they lie just outside the maximum extents of the groups specified by the user.

Clip To Zones submenu

The submenu contains items to select each of the saved clipping zones. . Selecting an item in this menu turns on clipping, if it wasn't already on, and restores clipping to the clipping region specified by the zone.

Save Clipping Zone

The current clipping region is used to add a new clipping zone or to update the clipping region associated with an existing clipping zone.

Edit Zones

Opens a dialog to edit the properties of zones. Zones may be renamed and the region stored with each zone may be edited. Zones may also be added and deleted via this dialog.

Masking Submenu

The Masking submenu commands are used to control the display of member in the frame. Masking allows you to define which members are visible and which are invisible.

No Masking

Turns off masking if it is on. This makes all members in the frame visible.

Mask Grey

Turns on masking and makes the masking mode grey. This means all members that have been masked out, will be drawn in grey.

Mask Invisible

Turns on masking and makes the masking mode invisible. This means all members that have been masked out, will be made invisible.

Mask To Frame

Turns on masking if it wasn't already on and makes all of the members in the frame visible.

Mask To Window

Turns on masking if it wasn't already on and makes all the members that lie completely inside the boundaries of the window visible.

Mask To Selection

Turns on masking and masks out all the members in the frame that are not selected. This means the only members in the frame that will be visible are those which were selected.

Mask To Group

Turns on masking and masks out all the members in the frame that are contained in the specified groups. This means the only members in the frame that will be visible are those which are in the selected groups.

Mask Out Selection

Turns on masking and masks out all the selected members in the frame. This has the effect of hiding the selected members and leaving all remaining visible members visible.

Mask Out Group

Turns on masking and masks out all the members in the frame contained in the specified groups. This has the effect of hiding the member in the selected groups and leaving all remaining visible members visible.

Current View Submenu

The view submenu commands allow you to view the frame from different angles.

Front

Orientates the view in the active window to the x-y plane with z-axis out of window.

Back

Orientates the view in the active window to the x-y plane with z-axis into the window.

Left

Orientates the view in the active window to the y-z plane with x-axis out of window.

Right

Orientates the view in the active window to the y-z plane with x-axis into the window.

Top

Orientates the view in the active window to the x-z plane with y-axis out of window.

Bottom

Orientates the view in the active window to the x-z plane with y-axis into the window.

3D

Sets a three dimensional view of the frame in the active window. Viewing angles can be modified using the angle slides on the bottom and right hand side of the window.

Toolbar Submenu

The Toolbar submenu commands allow you to make selected toolbars visible or invisible.

File Toolbar

Makes the File toolbar visible or invisible. The File toolbar contains common file functions.

View Toolbar

Makes the View toolbar visible or invisible. The View Toolbar provides common view functions such as Zoom and Pan.

Symbols Toolbar

Makes the Symbols toolbar visible or invisible. The Symbols Toolbar provides buttons to toggle the display of commonly used symbols.

Format Toolbar

Makes the Format toolbar visible or invisible. The Format Toolbar allows you to change the style of the font used in each of the windows in Multiframe.

Window Toolbar

Makes the Window toolbar visible or invisible. The Window Toolbar provides a quick way to move from window to window.

View3D Toolbar

Makes the View3D toolbar visible or invisible. The View3D Toolbar provides a quick way to move between 2D and 3D views in the graphics windows.

Group Toolbar

Makes the Group toolbar visible or invisible. The Group toolbar provides a simple way to transverse the different groups sets.

Rendering Toolbar

Makes the Rendering toolbar visible or invisible. The Rendering Toolbar allows you to modify the setting used in OpenGL rendering.

Clipping Toolbar

Makes the Clipping toolbar visible or invisible. The Clipping Toolbar allows you to enable or disable clipping, clip to a model and to clip to a stored clipping zone.

Generate Toolbar

Makes the Generate toolbar visible or invisible in the Frame Window. The Generate Toolbar allows you generate frames.

Joint Toolbar

Makes the Joint toolbar visible or invisible in the Frame Window. The Joint Toolbar allows you restrain joints.

Member Toolbar

Makes the Member toolbar visible or invisible in the Frame Window. The Member Toolbar allows you perform functions such as Add Member, Delete Member, and Sub-divide Member.

Geometry Toolbar

Makes the Geometry toolbar visible or invisible in the Frame Window. The Geometry Toolbar allows you perform functions such as rotating, duplicating and shearing part of a frame.

Drawing Toolbar

Makes the Drawing toolbar visible or invisible in the Frame Window. The Drawing Toolbar allows you perform functions such as adding and deleting members. It also allows the user to control various settings related to snapping and drawing.

Drawing load panel Toolbar

Makes the Drawing load panel toolbar visible or invisible in the Frame Window. The Drawing Load Panel Toolbar allows you perform functions such as adding and rotating support edges.

Load Panel Symbol Toolbar

Makes the load panel symbol toolbar visible or invisible. The Load Panel Symbol Toolbar allows you perform functions such as displaying axes, numbers, labels, support edges, tributary areas, panel loads and simulated member loads etc.

Drawing patch Toolbar

Makes the Drawing patch toolbar visible or invisible in the Frame Window. The Drawing Patch Toolbar allows you perform functions such as adding, rotating and flipping local axes.

Load Patch Symbol Toolbar

Makes the patch symbol toolbar visible or invisible. The Patch Symbol Toolbar allows you perform functions such as displaying internal meshes, edge nodes, internal nodes, colour fill, edges, axes, numbers, labels, materials, thickness, patch loads and plate axes, numbers, labels, materials, thickness and loads etc.

Actions Toolbar

Makes the Actions toolbar visible or invisible in the Plot Window. The Actions Toolbar allows you to view results of an analysis such as the Mz' and Deflection.

Load Case Toolbar

Makes the Load Case toolbar visible or invisible. The Loadcase toolbar provides a simple way to transverse the different load cases.

Load Toolbar

Makes the Load toolbar visible or invisible. The Load Toolbar provides common loading options.

Select Menu

The Select menu has commands for automatically selecting parts of the structure.

All

(Ctrl+A)
Automatically selects all the members in the frame.

Joints

Allows you to select a joint by number.

Joint Labels

Selecting a member using the joint label.

Linked Joints

Selects the members in specified linked joint groups.

Sections

Allows you to select all the members in the Frame window which have a given section type. You can choose the section type to be selected from a list of sections in the Sections Library.

Members

Allows you to select a member by number.

Member Labels

Selecting a member using the member label.

Member Slope

The Member Slope menu contains commands to select members aligned in various directions.

- **Horizontal**
  Automatically selects all the horizontal members (beams) in the frame.
- **Vertical**
  Automatically selects all the vertical members (columns) in the frame.
- **Sloping**
  Automatically selects all the vertical members (columns) in the frame.

Member Actions

Allows you to search through the structure to find members with actions, deflections or stresses in excess of limits you may enter. For example, you could find and select all of the members with a tensile stress greater than 21ksi.

Load Panels

Allows you to select a load panel by number.

Load Panel Labels

Selecting a load panel using the load panel label.

Patches

Allows you to select a patch by number.

Patch Labels

Selecting a patch using the patch label.

Plates

Allows you to select a plate by number.

Plate Labels

Selecting a plate using the plate label.

Design Members

Allows you to select a design member by number.

Design Member Labels

Selecting a design member using the member label.

Groups

Selects the members in specified groups.

From Window

The From Window menu contains commands to select members in the current window based upon the selection in another window.

- **Frame**
  Selects the members selected in the Frame Window.
- **Load**
  Selects the members selected in the Load Window.
- **Plot**
  Selects the members selected in the Plot Window.
- **Data**
  Selects the members selected in the Data Window.
- **Result**
  Selects the members selected in the Result Window.

Geometry Menu

The Geometry menu provides functions for creating and editing the geometry of a frame.

Add Member

(Insert)
Add a member to the structure in the Frame window. Press the mouse button to position the first joint of the member and drag to the position of the second joint and release the button. Windows users have the option of just clicking on the first joint, then clicking on the second joint position to draw a member.

Add Connected Members

(Shift+Insert)
Add a series of members to the structure in the Frame window. Press the mouse button to position the first joint of the first member and then move to the position of the second joint and press the button again. Further members can then be created by moving to the end position and clicking the mouse button. Press escape when you have finished adding members.

Delete Member

(Delete/Backspace)
Delete the selected members in the Frame window from the structure.

Add Spring Member

Adds a spring member to the structure in the Frame window. Press the mouse button to position the first joint of the member and drag to the position of the second joint and release the button. Windows users have the option of just clicking on the first joint, then clicking on the second joint position to draw a member.

Subdivide Member

(Ctrl B)
Divide all selected members in the Frame window into a number of equally sized smaller members.

Convert Member to Arc

Convert the selected members in the Frame window into a number of equally sized smaller members in the shape of an arc.

Add Rect Load Panel

Add a rectangle load panel to the structure in the Frame window. Position the first joint of the load panel to do a mouse click, and then position the second joint to do the second mouse click.

Add 4-node Load Panel

Add a quadrilateral load panel to the structure in the Frame window. Position the first joint of the load panel to do a mouse click, and then position the second joint to do the second mouse click, and then position the third joint to do the third mouse click, and then position the last joint to do the fourth mouse click.

Add 3-node Load Panel

Add a triangle load panel to the structure in the Frame window. Position the first joint of the load panel to do a mouse click, and then position the second joint to do the second mouse click, and then position the last joint to do the third mouse click.

Deleted Load Panel

Delete the selected load panels in the Frame window from the structure.

Add Rect Patch

Add a rectangle patch to the structure in the Frame window. Position the first joint of the patch to do a mouse click, and then position the second joint to do the second mouse click.

Add N-node Patch

Add an N-node patch to the structure in the Frame window. Position the first joint of the patch to do a mouse click, and then position the second joint to do the second mouse click, and then so on, position the next joint to do the next mouse click, and then position the last joint to do the last mouse click.

Generate Patches

To automatically add Patches between all of the members in a planar region (eg a floor or frame) in the Frame window, select the members which define the planar region, and then choose auto-generate patches command from the Geometry menu.

Rotate Patch Axes

Allow you to rotate the patch axes anti-clockwise;

Flip Patch z-axis

Allow you to flip patch z-axis;

Convert Load Panels to Patches

To automatically convert load panels to patches in the Frame window, select load panels, and then choose auto-convert patches command from the Geometry menu.

Deleted Patch

Delete the selected patches in the Frame window from the structure.

Deleted Plate

Delete the selected plates in the Frame window from the structure.

Duplicate

(Ctrl D)
Duplicates all the selected members in the frame a given number of times in a specified direction. A dialog allows you to enter the spacing in each direction and whether the duplicated members should be connected to the existing frame.

Rotate

Rotates all the selected joints in the frame a specified number of degrees about a specified axis. A dialog allows you to enter the number of degrees and the centre of rotation.

Rescale

Multiplies the coordinates of all the selected joints in the frame by a specified scaling factor. This has the effect of rescaling the structure by the specified amount.

Extrude

Creates new members from all the selected joints in the frame in a specified direction. Usually used to generate columns from a drawn floor plan or to generate beams out from an existing column line.

Move

(Ctrl M)
Allows you to either move the selected joints in the frame a specified distance or to move the origin to a new location.

Mirror

Allows you to mirror the selected members in the Frame window about a specified axis.

Shear

Allows you to shear the selected members in the Frame window in a specified direction and to a specified angle.

Generate

Allows you to automatically generate a continuous beam, portal frame, bay and story frame or curved beam.

Renumber

Allows you to automatically renumber the joints and/or members in the structure. This is convenient for sorting joint and member numbers by direction after you have been making modifications to the frame's geometry.

Advanced

The Advanced menu contains the following infrequently used commands.

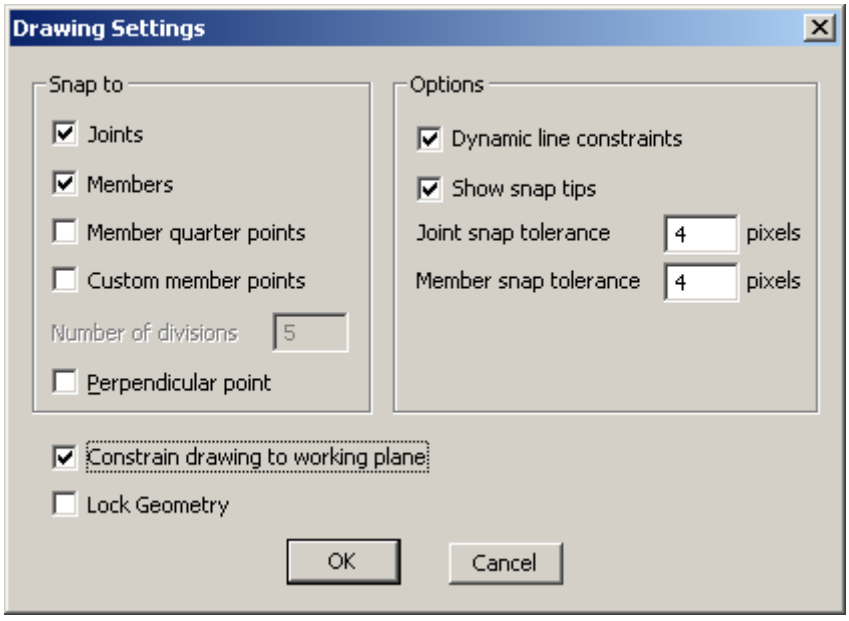
- **Add Joint**
  Adds a node at a specified location.
- **Merge Close Joints**
  Scans through the selected nodes in the Frame window and joins together any members which end at coincident nodes.
- **Align Joints**
  Aligns the nodes selected nodes in the Frame window to a specified position.
- **Intersect Members**
  Subdivides and connects all intersecting members in the current selection.
- **Merge Members**
  Merges groups of members in the current selection into a single member. Members to be merged into a single member are identified by finding all members that have the same section size, are collinear, are rigidly connected, and have the same member component.
- **Reverse Member Ends**
  Reverses the direction of the selected members local x-axes by swapping the nodes that define the ends of the members.
- **Disconnect Members**
  Disconnects the selected members from the remainder of the frame.

Drawing Settings

Multiframe has a number of options to control the behaviour of the cursor when drawing new members or dragging part of a frame. Most of these settings affect how the cursor snaps to objects in the model.

To modify the drawing and snapping settings

- **Choose Drawing Settings… from the Geometry menu**



- **Choose the options in the Snap To group box to specify how the cursor will snap to the existing joints and members.**

This includes options for snapping to quarter points of members and for specifying an arbitrary number of divisions. A number of options that control what happens when snapping to objects in a frame may also be specified.

- **Select the Dynamic line constraints option to automatically constrain the cursor to move along specified directions after snapping to joints.**
- **Selecting the option to Show Snap Tips will display a tool tip identifying what the cursor has snapped to.**
- **Enter the distance in pixels that the cursor will move to snap to a joint**
- **Enter the distance in pixels that the cursor will move to snap to a member. Generally this should be less then the joint snap tolerance.**

An important option for working with 3D models is to be able to constrain drawing to the drawing depth. When drawing new members this will force all members to be added at the current drawing depth. The cursor will still snap to points away from the drawing depth but the position of the mouse in 3D space will be computed by projecting a location back to the drawing plane defined by the drawing depth.

The last control in the dialog provides an option to lock the geometry and topology of the frame so that it cannot be modified.

- **Click the OK button**

Dynamic Line Constraints is a feature in Multiframe that allows drawing in any direction in a 3D view and for helping to align the model accurately. When moving the mouse from a joint, if the mouse is initially moved along the direction of one of the global axes then drawing will be constrained to that axis in 3D space. Furthermore, if the mouse is moved in the direction of a member attached to the joint then drawing will be constrained to the direction of that member in 3D space.

Many of these options can be access via the Drawing Toolbar.

Group Menu

The Group menu provides commands for organising the members in the structural model into groups or assemblies.

Create Design Member

Group the selected members together to form a multi-member design member.

Remove Design Member

Delete or split the selected members from multi-member design member(s).

Add Group

Adds a new group to the current group set using the selection within the window.

Edit Group

Edits the properties of a group.

Delete Group

Delete a group from the current group set.

Add to Group

Adds the selection as part of a group within the current group set.

Remove from Group

Removes the selection from a group within the current group set.

Add Group Set

Adds a new group set to the model.

Edit Group Set

Edits the properties of a group set.

Delete Group Set

Delete a group set from the model.

Current Group Set submenu

The submenu contains items to select the current group set. The current group set is the set displayed in the user interface.

Frame Menu

The Frame menu provides functions for editing the properties of a frame and it's components.

Joint Restraint

Restrain all the joints selected in the Frame window. A dialog box will appear allowing you to specify which type of restraint to apply.

Joint Mass

(Multiframe 4D only)
Additional mass added to a structure to add inertia for dynamic analysis.

Joint Spring

Prescribe a spring at all the joints selected in the Frame window. A dialog box will appear which allows you to specify the direction and stiffness of the spring.

Joint Type

Set the selected joint type as rigid or pinned. Rigid joints transmit moment and pinned joints do not.

Joint Linking

Links a group of joints together so that they move together in response to static or dynamic loads.

Joint Labels

Allows you to edit the user defined label associated with each joint.

Joint Orientation

Set the orientations of the local axes at the selected joints.

Section Type

Set the section type for all the members selected in the Frame window. A dialog box will appear which allows you to choose the section to use for the members selected.

Member Releases

Set the type of the selected members in the Frame window - members can be fixed or pinned at either or both ends.

Member Shear Area

Allows you to turn on or off the use of the optional calculation of deflection due to shear deformation.

Member Orientation

Set the orientation of the section for all the members selected in the Frame window. A dialog box will appear which allows you to rotate the section to the desired orientation.

Member Type

Allows you to choose whether a member is standard, tension only or compression only.

Member Offsets

(Multiframe 3D and 4D only)
Allows you specify the size of rigid offset connections at the ends of the member.

Member Masses

(Multiframe 4D only)
Specify whether the mass of the selected members should be included when a dynamic analysis is performed. This does not affect the inclusion of a member's weight when a static analysis is carried out using self weight. That is controlled by the Member Self Weight item from the Load menu.

Member Labels

Allows you to edit the user defined label associated with each member.

Member Component

Allows you to specify what type of component in the model that the member represents.

Member End Springs

This command is used to specify semi-rigid behaviour at the ends of the member Any of the members degrees of freedom may be specified as semi-rigid and a suitable stiffness spring stiffness associated with the connection.

Spring Member Stiffness

Sets the stiffness of the spring members selected in the Frame Window.

Load Panel Colours

Allows you to edit colours of the selected load panels.

Load Panel Labels

Allows you to edit labels of the selected load panels.

Load Panel Supports

Allows you to edit supports of the selected load panels.

Patch Meshing

Allows you to change mesh densities of the selected patches.

Patch Release

Allows you to release one or all edges of the selected patches.

Patch Colours

Allows you to edit colours of the selected patches.

Patch Labels

Allows you to edit labels of the selected patches.

Patch Thickness

Allows you to edit thickness of the selected patches.

Patch Materials

Allows you to edit materials of the selected patches.

Rotate Patch Axes

Allows you to rotate patch axes anti-clockwise.

Flip Patch z-axis

Allows you to flip patch z-axis.

Patch Reference Axes

Allows you to set a reference axes for all selected patches.

Plate Release

Allows you to release one or all edges of the selected plates.

Plate Colours

Allows you to edit colours of the selected plates.

Plate Labels

Allows you to edit labels of the selected plates.

Plate Thickness

Allows you to edit thickness of the selected plates.

Plate Materials

Allows you to edit materials of the selected plates.

Load Menu

All commands available in the Load menu act on the current load case.

Unload Joint

Remove all loads from the selected joints in the Load window.

Global Joint Load

Add a point load to each of the selected joints in the Load window. The load will be aligned with the global coordinate system. A dialog box will appear allowing you to specify a direction and magnitude for the load.

Global Joint Moment

Add a point moment to each of the selected joints in the Load window. The load will be aligned with the global coordinate system. A dialog box will appear allowing you to specify a direction and magnitude for the moment.

Local Joint Load

Add a point load to each of the selected joints in the Load window. The load will be aligned with the local orientation of the joint. A dialog box will appear allowing you to specify a direction and magnitude for the load.

Local Joint Moment

Add a point moment to each of the selected joints in the Load window. The load will be aligned with the local orientation of the joint. A dialog box will appear allowing you to specify a direction and magnitude for the moment.

Prescribed Displacement

Prescribe a displacement at all the joints selected in the Frame window. A dialog box will appear which allows you to specify the direction and magnitude of the displacement.

Unload Member

Remove all loads from the selected members in the Load window.

Global Dist'd Load

Add a distributed load to each of the selected members in the Load window. A dialog box will appear which allows you to specify the magnitude, direction and position of the load. A global distributed load acts parallel to one of the global x, y or z axes.

Global Point Load

Add a point loads to each of the selected members in the Load window. A dialog box will appear which allows you to specify the magnitude, direction and position of the load. A global point load acts parallel to one of the global x, y or z axes.

Global Moment

Add a point moment to each of the selected members in the Load window. A dialog box will appear which allows you to specify the magnitude, direction and position of the load for the members selected. A global moment acts about one of the global x, y or z axes.

Local Dist'd Load

Add a local distributed load to each of the selected members in the Load window. A dialog box will appear which allows you to specify the magnitude, direction and position of the load. A local distributed load acts parallel to one of the local x', y' or z' member axes.

Local Point Load

Add a local point load to each of the selected members in the Load window. A dialog box will appear which allows you to specify the magnitude, direction and position of the load. A local point load acts parallel to one of the local x', y' or z' member axes.

Local Moment

Add a local point moment to each of the selected members in the Load window. A dialog box will appear which allows you to specify the magnitude, direction and position of the load. A local moment acts about one of the local x', y' or z' member axes.

Thermal Load

Add a thermal load to each of the selected members in the Load window. A dialog box will appear which allows you to specify the temperature, thermal coefficient, depth and direction of the load. A thermal load may either be uniform or a linear variation of temperature through the depth of the member.

Member Self Weight

Specify whether the self weight of the selected members should be included or ignored during a static analysis that includes a self weight load case. Note that this does not affect the inclusion of member masses in a dynamic analysis, which is controlled by the Member Mass command from the Frame menu.

Patch Self Weight

Specify whether the self weight of the selected patches should be included or ignored during a static analysis that includes a self weight load case.

Plate Self Weight

Specify whether the self weight of the selected plates should be included or ignored during a static analysis that includes a self weight load case.

Dynamic Load

(Multiframe 4D only)
Allows you to apply a dynamic time varying load to selected joints in the Load window.

Unload Load Panel

Remove all loads from the selected load panels in the Load window.

Global Panel Load

Add a pressure load to each of the selected load panels in the Load window. A dialog box will appear which allows you to specify the pressure magnitude and direction. A global distributed load acts parallel to one of the global x, y or z axes.

Local Panel Load

Add a local pressure load to each of the selected load panels in the Load window. A dialog box will appear which allows you to specify the pressure magnitude and direction of the load. A local distributed load acts parallel to one of the local x', y' or z' member axes.

Unload Patch

Remove all loads from the selected patches in the Load window.

Global Patch Point Load

Add a point load to each of the selected patches in the Load window. A dialog box will appear which allows you to specify the magnitude, locations and direction. A global point load acts parallel to one of the global x, y or z axes.

Local Patch Point Load

Add a local point load to each of the selected patches in the Load window. A dialog box will appear which allows you to specify the magnitude, locations and direction of the load. A local distributed load acts parallel to one of the local x', y' or z' member axes.

Global Patch Load

Add a pressure load to each of the selected patches in the Load window. A dialog box will appear which allows you to specify the magnitude and direction. A global point load acts parallel to one of the global x, y or z axes.

Local Patch Load

Add a local pressure load to each of the selected patches in the Load window. A dialog box will appear which allows you to specify the pressure magnitude and direction of the load. A local distributed load acts parallel to one of the local x', y' or z' member axes.

Unload Plate

Remove all loads from the selected plates in the Load window.

Global Plate Load

Add a pressure load to each of the selected plates in the Load window. A dialog box will appear which allows you to specify the pressure magnitude and direction. A global distributed load acts parallel to one of the global x, y or z axes.

Local Plate Load

Add a local pressure load to each of the selected plates in the Load window. A dialog box will appear which allows you to specify the pressure magnitude and direction of the load. A local distributed load acts parallel to one of the local x', y' or z' member axes.

Display Menu

The Display menu contains commands for controlling which data is displayed in the different windows.

Symbols

This command brings up a dialog box that allows you to specify which symbols will be displayed in the Frame, Load and Plot windows. You can turn on or off the display of joint and member numbers, loads, restraints, member axes, masses and names of sections.

Legend

The Legend menu contains commands to control the appearance of legends.

- **Top Left**
  Display the legend in the top left-hand corner of the window.
- **Top Right**
  Display the legend in the top right-hand corner of the window.
- **Bottom Left**
  Display the legend in the bottom left-hand corner of the window.

- **Bottom Right**
  Display the legend in the bottom right-hand corner of the window.
- **Title Font**
  Sets the font size and style used display the title of the legend.
- **Item Font**
  Sets the font size and style used display the items listed within the legend.
- **Item Colour**
  Specify the colour associated with the items listed within the legend.

Data

See the "Data Submenu"

Results

See the "Results Submenu"

Actions

See the "Actions Submenu"

Stresses

See the "Stresses Submenu"

Patch Internal Forces

See the "Patch Internal Force Submenu"

Patch Stresses

See the "Patch Stresses"

Deflection

Display the deflection for the current load case in the Plot window. If the current load case is a dynamic case, the deflection will represent a mode shape of the structure.

Animate

Animate the diagram in the front window. In the Frame and Load windows, this command only operates in the 3D view and will display views of the structure at a range of viewing angles. Once the diagrams have been displayed you can animate the range of views by moving the mouse back and forth. In the Plot window this command will compute a series of diagrams showing the change as loading increases from zero to its prescribed level.

If you turn on the Save To Animation File check box in the Animate dialog, the animation will be saved into an AVI movie file. The movie is displayed in the window with the standard AVI controller and you can play it if you wish using the standard Windows Media Player program.
QuickTime.

Render

Display the frame complete with web and flange details. This display will help you visualise the orientation and section types for the frame. The rendering is not an exact display of the actual shape of the members in the structure but more a visual guide to the relative size and orientation of the sections in the frame.

You can interrupt the drawing of a rendered view of the structure by pressing the escape key.

Plot

Specify the precision to be used in the display of deflection diagrams in the Plot window, which action, if any, is to be overlaid onto the deflection diagram and what scaling factor should be applied to the current diagram.

The overlaid action will be displayed as a colour on the deflected shape. Red indicates a high value of the action relative to the rest of the structure while blue indicates a relatively low value.

Plot Colours

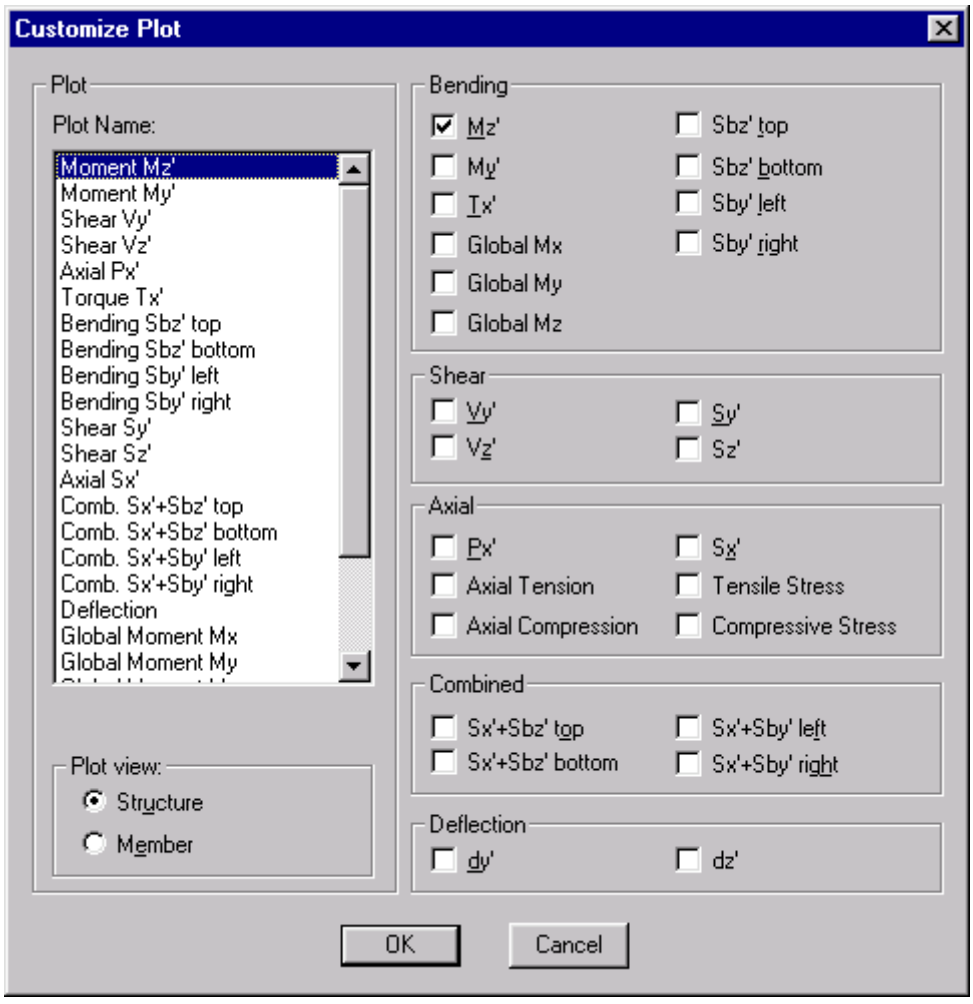
Specify which colours to use for the various diagrams.

Customise Plot

You can customise the display of diagrams in the Plot window to allow the display of one or more diagrams simultaneously. This applies to both global and local diagrams.

To choose which diagrams are displayed in the Plot window

- **Choose Customise Plot… from the Display menu**



A dialog will appear listing the menu items controlling the plot display and the actions, stresses and deflections which can be displayed.

- **Click on the name of the menu item you wish to change**
- **Click on the Member button**

- **Set the check boxes of the actions you wish to display in the local member diagrams**
- **Click on the Structure button**
- **Set the check boxes of the actions you wish to display on the diagram of the whole structure**
- **Click the OK button**

Data Submenu

The Data submenu controls which table of data, will be displayed in the Data window. One table can be displayed and edited at a time.

Joints

Display a table of joint coordinates.

Member Geometry

Display a table of member data describing the geometry of the members in the frame. This data includes joint numbers at the ends, type, member type, length, slope and orientation. offsets connection dimensions.

Member Properties

Display a table of member property data including section type, orientation, member type, and member releases.

Member End Properties

Display a table of property associated with the end of a member including member offsets and member releases.

Member End Springs

Display a table of member end springs, and their properties, which represent semi-rigid connections in the model.

Joint Loads

Display a table of joint load positions and magnitudes.

Prescribed Displacements

Display a table of joint load positions and magnitudes.

Member Loads

Display a table of member load positions and magnitudes.

Thermal Loads

Display a table of thermal load magnitudes.

Restraints

Display a table of joint restraints and prescribed displacements.

Linked Joints

Links a group of joints so that they move together in response to static or dynamic loads.

Springs

Display a table of joint spring stiffness’s.

Spring Members

Display a table of spring members.

Joint Masses

(Multiframe 4D only)
Display a table of joint masses.

Sections

Display a table of sections summarizing the use of sections in the frame. The table includes the number, length and mass of each type of section used.

Materials

Display a table of materials summarizing the use of materials in the frame. The table includes the name, group, type and density of each type of material used.

Step Loads

(Multiframe 4D)
Display a table of dynamic joint load positions and associated tine series.

Load Cases

Display a table of static load cases.

Load Panel Geometry

Display a table of load panel data describing the geometry of the load panels in the frame. This data includes panel numbers, label, side elements, and colour etc.

Load Panel Loads

Display a table of load panel data describing the loads of the load panels. This data includes panel numbers, label, load shape, direction and magnitude etc.

Patch Geometry

Display a table of patch data describing the geometry of the patches in the frame. This data includes patch numbers, label, mesh type, material, thickness and colour etc.

Patch Edge Release

Display a table of patch data describing the edge releases of the patches. This data includes patch numbers, label, sides and release details.

Patch Loads

Display a table of patch data describing the loads of the patches. This data includes patch numbers, label, load shape, direction and magnitude etc.

Patch Openings

Display a table of patch opening data describing the geometry of the openings in the frame. This data includes patch opening numbers, label, corner nodes, patch belonging to etc.

Patch Constrains

Display a table of patch line constrains. This data includes constrain numbers, label, patch belonging to, nodes, type of constrain & line constrain from etc.

Plate Geometry

Display a table of plate data describing the geometry of the plates in the frame. This data includes plate numbers, label, material, thickness and colour etc.

Plate Release

Display a table of plate data describing the edge releases of the plates. This data includes plate numbers, label, joint number and release details.

Plate Loads

Display a table of plate data describing the loads of the plates. This data includes plate numbers, label, load shape, direction and magnitudes etc.

Analysis Settings

Display a table summarizing the setting used in the nonlinear analysis of each load case.

Results Submenu

The items in the Results sub-menu allow you to specify which table of results should be displayed in the Results window.

Displacements

Display the computed joint displacements for the current load case in the Result window.

Reactions

Display the computed joint reactions for the current load case in the Result window.

Member Actions

Display the computed member actions for the current load case in the Result window.

Max Member Actions

Display the maximum actions on members for the current load case in the Result window.

Spring Member Actions

Display the actions within spring members for the current load case in the Result window.

Member Stresses

Display the computed stresses at the end of each member for the current load case in the Result window.

Max Member Stresses

Display the maximum stresses on members for the current load case in the Result window.

Member Details

Display the member actions and stresses at the number of points along a selected member.

Member Buckling

(Multiframe 3D and 4D only)
Display the results of buckling analysis which includes member axial forces and effective lengths and effective length factors.

Patch Internal Forces

Display the patch interaction forces within patches for the current load case in the Result window.

Patch Stresses

Display the stresses within patches for the current load case in the Result window.

Natural Frequencies

(Multiframe 4D only)
Display the computed frequencies, periods, participation factors, participating mass ratios and modal masses for each of the mode shapes determined by a modal analysis.

Actions Submenu

The items in the Actions submenu may be used to control which type of force or moment is displayed in the Plot window.

Moment Mz'

Display the computed bending moments about the local z' axis for the current load case in the Plot window.

Moment My'

Display the computed bending moments about the local y' axis for the current load case in the Plot window.

Shear Vy'

Display the computed shear forces in the local y' direction for the current load case in the Plot window.

Shear Vz'

Display the computed shear forces in the local z' direction for the current load case in the Plot window.

Axial Px'

Display the computed axial forces for the current load case in the Plot window.

Axial Tension

Display the computed axial tensile forces for the current load case in the Plot window.

Axial Compression

Display the computed axial compressive forces for the current load case in the Plot window.

Torque Tx'

Display the computed torque about the local x' axis for the current load case in the Plot window.

Global Mx

Display the computed bending moments about the global x axis for the current load case in the Plot window.

Global My

Display the computed bending moments about the global y axis for the current load case in the Plot window.

Global Mz

Display the computed bending moments about the global z axis for the current load case in the Plot window.

Stresses Submenu

The items in the Stresses submenu may be used to control which type of stress is displayed in the Plot window.

Bending Sbz' top

Display the computed bending stress about the local z' axis at the top of each member for the current load case in the Plot window.

Bending Sbz' bottom

Display the computed bending stress about the local z' axis at the bottom of each member for the current load case in the Plot window.

Bending Sby' left

Display the computed bending stress about the local y' axis at the left of each member (as viewed from Joint 2) for the current load case in the Plot window.

Bending Sby' right

Display the computed bending stress about the local y' axis at the right of each member (as viewed from Joint 2) for the current load case in the Plot window.

Shear Sy'

Display the computed shear stress in the local y' direction for the current load case in the Plot window.

Shear Sz'

Display the computed shear stress in the local z' direction for the current load case in the Plot window.

Axial Sx'

Display the computed axial stress for the current load case in the Plot window.

Tensile Sx'

Display the computed axial tensile stress for the current load case in the Plot window.

Compressive Sx'

Display the computed axial compressive stress for the current load case in the Plot window.

Comb Sx' + Sbz' top

Display the combined axial stress and bending stress about the local z' axis at the top of each member.

Comb Sx' + Sbz' bottom

Display the combined axial stress and bending stress about the local z' axis at the bottom of each member.

Comb Sx' + Sby' left

Display the combined axial stress and bending stress about the local y' axis at the left of each member.

Comb Sx' + Sby' right

Display the combined axial stress and bending stress about the local y' axis at the right of each member.

Patch Internal Forces Submenu

The items in the Patch Internal Forces submenu may be used to control which type of internal force is displayed in the Plot window.

Moment Mxx'

Display the computed plate bending moments about the local xx' axis for the current load case in the Plot window.

Moment Myy'

Display the computed plate bending moments about the local yy' axis for the current load case in the Plot window.

Moment Mxy'

Display the computed plate twisting moments in x-y' plane for the current load case in the Plot window.

Shear Vxz'

Display the computed plate transverse shear force about the local z' axis in x-z' plane for the current load case in the Plot window.

Shear Vyz'

Display the computed plate transverse shear force about the local z' axis in y-z' plane for the current load case in the Plot window.

Moment Mmin

Display the computed minimum plate bending moments about the principal axis for the current load case in the Plot window.

Moment Mmax

Display the computed maximum plate bending moments about the principal axis for the current load case in the Plot window.

Force Fxx'

Display the computed membrane direct forces about the local xx' axis for the current load case in the Plot window.

Force Fyy'

Display the computed membrane direct forces about the local xx' axis for the current load case in the Plot window.

Force Fxy'

Display the computed membrane shear forces in x-y' plane axis for the current load case in the Plot window.

Force Fmin

Display the computed minimum membrane forces about the principal axis for the current load case in the Plot window.

Force Fmax

Display the computed maximum membrane forces about the principal axis for the current load case in the Plot window.

Patch Stresses Submenu

The items in the Patch Stresses submenu may be used to control which type of patch stress is displayed in the Plot window.

Sxx'

Display the computed in-plane direct stresses about the local xx' axis for the current load case in the Plot window.

Syy'

Display the computed in-plane direct stresses about the local yy' axis for the current load case in the Plot window.

Szz'

Display the computed transverse direct stresses about the local zz' axis for the current load case in the Plot window.

Sxy'

Display the computed in-plane shear stresses in x-y' plane for the current load case in the Plot window.

Sxz'

Display the computed transverse shear stresses in x-z' plane for the current load case in the Plot window.

Syz'

Display the computed transverse shear stresses in yz' plane for the current load case in the Plot window.

Von Mises

Display the computed Von Mises stresses for the current load case in the Plot window.

Tresca

Display the computed Tresca stresses for the current load case in the Plot window.

Case Menu

The items in the Case menu allow you to work with load cases.

Add Case

Add a new load case - enter load factors if desired. The Add Case menu is a sub menu of the Case menu, and is described in detail following this section on the Case menu.

Edit Case

The Edit Case command allows you to modify the attributes of the current load case.

Reorder Cases

The Reorder Cases command allows you to sort the order of load cases in the list.

Delete Case

The Delete Case command allows you to delete one or more load cases.

Linear Results

This command displays the linear results of the current load case.

Nonlinear Results

This command displays the nonlinear results of the current load case.

Mode Shapes submenu

(Multiframe 4D)
The submenu contains items to select each of the modes shapes computed by a modal analysis. The results associated with selected mode shape will be displayed within the application. This affects the display of deflections in the Plot and Result windows.

Load Case *xxx* menu items

Display and allow editing of loads associated with the selected load case in the Load and Data windows. Display results for the selected load case in the Plot and Result windows.

Load Case

(Ctrl L)
Dialog box that allows you to choose a load case.

Add Case Submenu

The Add Case submenu allows you to enter load factors if desired.

Self Weight:

Adds a Self Weight Load Case

Static

Adds a Static Load Case

Static Combined

Combines and factors several load cases

Envelope

Adds an Envelope Load Case

Time History

(Multiframe 4D)
Adds a Time History Load Case

Seismic

(Multiframe 4D)
Adds a Seismic Load Case

Analyse Menu

Linear

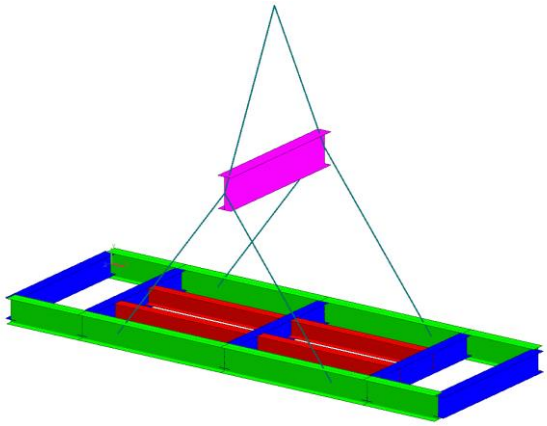
Linear Analysis

Nonlinear

Nonlinear Analysis

Buckling

A new type of analysis has been added for analysing frames which are used for lifting calculations. Typical examples would be lifting of baskets and skids with slings which are attached to a crane.



Such models are typified by a lack of restraint in the horizontal plane which allows the frame to either rotate or swing freely. If you model a frame like this you will often get the error message “Solution does not make sense”. The lifting analysis function automatically adds additional spring restraints to the model to provide sufficient horizontal restraint to allow the model to be analysed. This eliminates the need for you to apply your own manual spring restraints.

The stiffness of these restraints is automatically set by Multiframe so as to have a minimal effect on the actions within the frame. They are set at a small proportion of the other stiffnesses in the frame. While you are creating your structural model, you should review the location of the centre of mass and ensure that the lifting point is located as near as possible to a location directly above the centre of mass of the structure and its loads.

Note also that when modelling slings or lifting cables you should release both moments and torsion at the ends of those members. If you release all of the sling members connecting to the lifting point, be sure to applied a fixed restraint to the lifting point to prevent that node from rotating.

A lifting analysis is performed via the Lifting… command in the Analysis menu. A lifting analysis is in fact a special type of linear analysis and as such the results of the lifting analysis are stored as the results of a linear analysis. If your sling members are set to Tension-Only, then a dialog will appear allowing you to take those effects into account during analysis. You are no longer required to use a NonLinear analysis for Tension-Only effects.

Some care must be taken in performing a lifting analysis as the additional spring restraints added for the analysis will also help to suppress structural mechanisms. After analysis is complete, you will notice that all nodes in the model will have reactions. You should see normal reactions at any nodes where you have applied a restraint, and at other nodes where automatic springs have been applied, you should see reactions with a small value. These small reactions are a consequence of the automatically applied springs. If you see unacceptably large reactions at any of these nodes, then you should use manual spring restraints and carry out a normal linear analysis.
Buckling Analysis

Modal

Modal Analysis

Time History

Time History Analysis

Batch Analysis

Batch Analysis

Calculate

(Alt =) Calculate the value of the expressions in the CalcSheet.

Time Menu

(Multiframe 4D)
The items in the Time menu control which increment is selected in a time history analysis.

Maximum Envelope

Displays a Maximum Envelope of all increments in the current time history analysis case.

Minimum Envelope

Displays a Minimum Envelope of all increments in the current time history analysis case.

Absolute Envelope

Displays an Absolute Envelope of all increments in the current time history analysis case.

Time History

(Ctrl H)
Set the currently displayed time history increment.

Window Menu

The Windows menu provides window specific functionality.

Cascade

Displays all the Windows behind the active Windows.

Tile Horizontal

Layout all visible windows across the screen.

Tile Vertical

Layout all visible windows down the screen.

Arrange Icons

Rearranges the icons of any minimised window so that they are collected together at the bottom of the Multiframe program window.

Editing Layout

Makes the Frame, Data and Load windows all visible at once. This is useful when you are defining the structure and its loads and restraints.

Result Layout

Makes the Plot, Result and Load windows all visible. This is useful when you are examining the results of the analysis.

Calcs Layout

Makes the Plot, Load and CalcSheet windows visible. This is useful when you are doing calculations for a given member.

Report Layout

Makes the Plot, Load and Report windows visible. This is useful when you are reviewing a summary report of design calculations.

Frame

Makes the Frame window visible and brings it to the front.

Data

Makes the Data window visible and brings it to the front.

Load

Makes the Load window visible and brings it to the front.

Result

Makes the Result window visible and brings it to the front.

Plot

Makes the Plot window visible and brings it to the front.

CalcSheet

Makes the CalcSheet window visible and brings it to the front.

Report

Makes the Report window visible and brings it to the front.

Help Menu

Provides access to an on-line help system.

Multiframe Help

This command launches the Multiframe help file.

Automation Help

This command launches the Multiframe Automation help file.

Automation Reference

This command launches the Multiframe Automation Reference file.

Steel Designer Help

This command launches the Steel Designer help file.

Online Support

This command starts up your web browser and opens the support page on the Formation Design Systems web site. This web page includes online version of the manual, release notes, knowledge base and more.

Check For Updates

This command opens the web browser and opens the web page on the Formation Design Systems web site where the latest version of Multiframe are listed. Users with current subscription can log in to the website and download the latest update.

Learn Multiframe

This command starts the Learning Multiframe introductory training document. After clicking on this command in the Help menu, Multiframe will check for a Learning Multiframe installation on your computer and, if detected, starts up the PDF document. If Learning Multiframe is not detected, Multiframe will start up the online version of Learning Multiframe in your web browser. Learning Multiframe can be installed directly from the installation CD or from a download from our website. More information can be found at: http://www.formsys.com/support/learning-centre/learning-multiframe.

About Multiframe

Tells which version of Multiframe you are using and how many joints, members and forces etc. there are in the structure.

Chapter 4 Multiframe Analysis

This chapter describes the analysis methods used in Multiframe:

Method of Analysis

This section describes the methods and conventions that Multiframe uses.

- Matrix Stiffness Method
- Axes and Sign Convention
- Member Actions
- Modal Analysis
- Capacity
- Nonlinear Analysis
- Plate Analysis

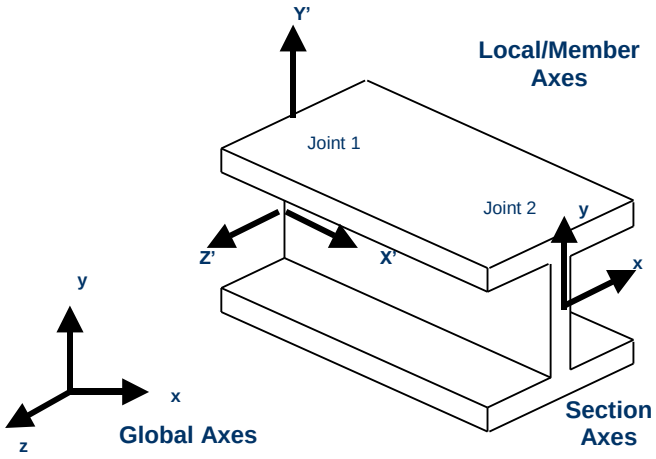
Matrix Stiffness Method

Multiframe uses the matrix stiffness method of solution for solving a system of simultaneous equations to determine the forces and deflections in a structure. Multiframe carries out a first order, linear elastic analysis to determine these forces and deflections. You should be familiar with the concepts and application of the matrix stiffness method before using this software.

The matrix stiffness method forms a stiffness matrix for each member of the structure and given a list of applied loading, solves a system of linear simultaneous equations to compute the deflections in the structure. The internal forces and reactions are then computed from these deflections. Multiframe does take into account deformations due to shear action in deep beams if the Member Shear option in the Frame menu has been turned on. Multiframe does not take into account deformations due to warping deformation due to torsion

Axes and Sign Convention

Multiframe uses two coordinate systems for defining geometry and loading. The global coordinate system is a right handed x, y, z system with y always running vertically and x and z running horizontally. Gravity loads due to self weight are always applied in the negative y direction. To distinguish between local and global axes, Multiframe uses the ' suffix to indicate a local axis.



Each member in the structure is defined by the two joints at its ends. The local coordinate system is a right handed x, y, z system with the x axis running along the member from joint 1 to joint 2. The direction of the y' axis is specified by your setting of the section orientation. The orientation is the angle between the y' axis and a vertical plane passing through the ends of the member measured from the vertical plane towards the y' axis as viewed from joint 2 looking towards joint 1.

Multiframe uses six degrees of freedom at each joint when performing its calculations (see the matrix on the next page). These comprise three displacements along the axes and three rotations about the axes at each joint.

The local element stiffness matrix K used by Multiframe is as follows:

$\frac{AE}{L}$	0	0	0	0	0	$\frac{-AE}{L}$	0	0	0	0	0
0	$\frac{12EIz'}{L^3}$	0	0	0	$\frac{6EIz'}{L^2}$	0	$\frac{-12EIz'}{L^3}$	0	0	0	$\frac{6EIz'}{L^2}$
0	0	$\frac{12EIy'}{L^3}$	0	$\frac{-6EIy'}{L^2}$	0	0	0	$\frac{-12EIy'}{L^3}$	0	$\frac{-6EIy'}{L^2}$	0
0	0	0	$\frac{GJ}{L}$	0	0	0	0	0	$\frac{-GJ}{L}$	0	0
0	0	$\frac{-6EIy'}{L^2}$	0	$\frac{4EIy'}{L}$	0	0	0	$\frac{6EIy'}{L^2}$	0	$\frac{2EIy'}{L}$	0
0	$\frac{6EIz'}{L^2}$	0	0	0	$\frac{4EIz'}{L}$	0	$\frac{-6EIz'}{L^2}$	0	0	0	$\frac{2EIz'}{L}$
$\frac{-AE}{L}$	0	0	0	0	0	$\frac{AE}{L}$	0	0	0	0	0

$$\begin{bmatrix}
0 & \frac{-12EIz'}{L^3} & 0 & 0 & 0 & \frac{-6EIz'}{L^2} & 0 & \frac{12EIz'}{L^3} & 0 & 0 & 0 & \frac{-6EIz'}{L^2} \\
0 & 0 & \frac{-12EIy'}{L^3} & 0 & \frac{6EIy'}{L^2} & 0 & 0 & 0 & \frac{12EIy'}{L^3} & 0 & \frac{6EIy'}{L^2} & 0 \\
0 & 0 & 0 & \frac{-GJ}{L} & 0 & 0 & 0 & 0 & 0 & \frac{GJ}{L} & 0 & 0 \\
0 & 0 & \frac{-6EIy'}{L^2} & 0 & \frac{2EIy'}{L} & 0 & 0 & 0 & \frac{6EIy'}{L^2} & 0 & \frac{4EIy'}{L} & 0 \\
0 & \frac{6EIz'}{L^2} & 0 & 0 & 0 & \frac{2EIz'}{L} & 0 & \frac{-6EIz'}{L^2} & 0 & 0 & 0 & \frac{4EIz'}{L}
\end{bmatrix}$$

A = Area, E = Young's Modulus, G= Shear Modulus, L = Length, J = Torsion Constant, I = Moment of Inertia

Where each element behaves according to the equation

F=Kx

Where F is the vector of applied loads, K is the stiffness matrix above and x is a vector of calculated displacements

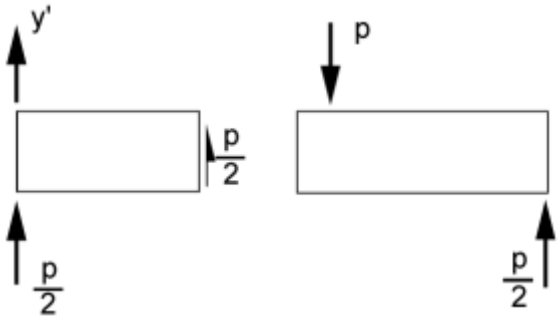
F={Px1, Py1, Pz1, Mx1, My1, Mz1, Px2, Py2, Pz2, Mx2, My2, Mz2}
x={dx1, dy1, dz1, Øx1, Øy1, Øz1, dx2, dy2, dz2, Øx2, Øy2, Øz2}

All relative to the local member coordinate system.

Note that the Iz' and Iy' above refer to the moment of inertia with respect to the local coordinate system. Since common engineering convention is to use Ix and Iy for the major and minor moments of inertia of a section, for a member with a section orientation of zero, Iz' corresponds to the conventional Ix while Iy' corresponds to the conventional Iy. The Multiframe Sections Library uses the conventional Ix, Iy notation for the two moments of inertia of the sections stored in the library.

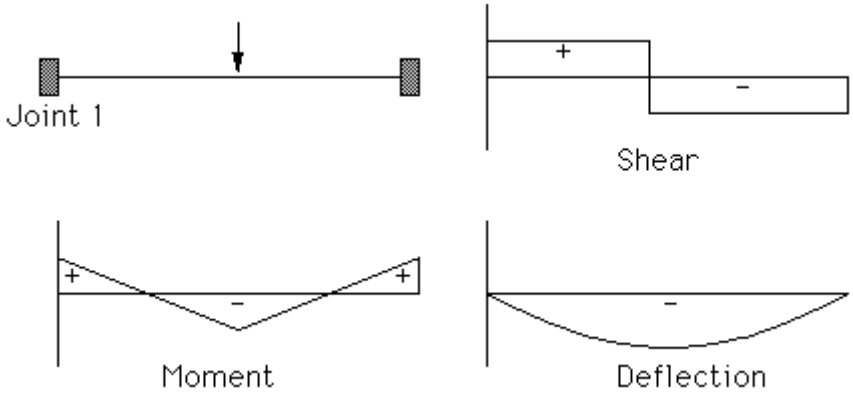
Member Actions

Multiframe computes member actions relative to the local member coordinate system. When calculating an action at an intermediate point along a member, Multiframe checks the free body diagram of the member to the left of the point of interest and uses the balance of forces at this point for the sign of the computed action. As an example, consider the shear force at a point on a simple beam subject to a central point load.



Following the above approach at the left hand portion of a beam, the sum of the shear forces is therefore positive as shown in the diagram above.

For the common case of a beam with joint 1 at the left hand end and joint 2 at the right hand end, a load acting downwards will be negative in magnitude and the forces will be as shown below.



When you specify the loads applied to a structure it will not be necessary to enter the signs of the load magnitudes since Multiframe will offer you a number of icons to choose from to specify a direction.

Modal Analysis

Multiframe4D allows you to perform a modal analysis of a frame. This analysis will determine the natural frequencies and mode shapes of the frame. These frequencies and mode shapes will reflect the interaction between the stiffness of the frame and the inertial effects of its mass and any joint masses you have applied to it.

Multiframe4D determines the natural dynamic response of the frame by using the subspace iteration method. This method solves the equation

$$[m]\{\ddot{u}\} + [k]\{u\} = \{0\}$$

Where [m] is the mass matrix, {ü} is the vector of joint accelerations, [k] is the stiffness matrix and {u} is the vector of joint displacements. The solutions of this equation of undamped free vibration, of which there are a number, represent the natural responses of the frame. Multiframe will calculate the solutions corresponding to the longest periods of vibration (i.e. the lowest frequencies).

The local member mass matrix used by Multiframe4D varies according to whether a lumped or distributed mass model is chosen in the Analysis dialog.

The mass matrix for a lumped model is as follows

$\frac{mL}{2}$	0	0	0	0	0	0	0	0	0	0	0
0	$\frac{mL}{2}$	0	0	0	0	0	0	0	0	0	0
0	0	$\frac{mL}{2}$	0	0	0	0	0	0	0	0	0
0	0	0	$\frac{mLIo}{2A}$	0	0	0	0	0	0	0	0
0	0	0	0	0	0	0	0	0	0	0	0
0	0	0	0	0	0	0	0	0	0	0	0
0	0	0	0	0	0	$\frac{mL}{2}$	0	0	0	0	0
0	0	0	0	0	0	0	$\frac{mL}{2}$	0	0	0	0
0	0	0	0	0	0	0	0	$\frac{mL}{2}$	0	0	0
0	0	0	0	0	0	0	0	0	$\frac{mLIo}{2A}$	0	0
0	0	0	0	0	0	0	0	0	0	0	0
0	0	0	0	0	0	0	0	0	0	0	0

A = Area, m = Mass per unit length, Io= polar moment of inertia=Ix + Iy, L = Length

The mass matrix for a distributed mass model (also known as the consistent mass matrix in some texts) is as follows

$\frac{140mL}{420}$	0	0	0	0	0	$\frac{70mL}{420}$	0	0	0	0	0
0	$\frac{156mL}{420}$	0	0	0	$\frac{22mL^2}{420}$	0	$\frac{54mL}{420}$	0	0	0	$\frac{-13mL^2}{420}$
0	0	$\frac{156mL}{420}$	0	$\frac{-22mL^2}{420}$	0	0	0	$\frac{54mL}{420}$	0	$\frac{13mL^2}{420}$	0
0	0	0	$\frac{140mLIo}{420A}$	0	0	0	0	0	$\frac{70mLIo}{420A}$	0	0

0	0	$\frac{-22mL^2}{420}$	0	$\frac{4mL^3}{420}$	0	0	0	$\frac{-13mL^2}{420}$	0	$\frac{-3mL^3}{420}$	0
0	$\frac{22mL^2}{420}$	0	0	0	$\frac{4mL^3}{420}$	0	$\frac{13mL^2}{420}$	0	0	0	$\frac{-3mL^3}{420}$
$\frac{70mL}{420}$	0	0	0	0	0	$\frac{140mL}{420}$	0	0	0	0	0
0	$\frac{54mL}{420}$	0	0	0	$\frac{13mL^2}{420}$	0	$\frac{156mL}{420}$	0	0	0	$\frac{-22mL^2}{420}$
0	0	$\frac{54mL}{420}$	0	$\frac{-13mL^2}{420}$	0	0	0	$\frac{156mL}{420}$	0	$\frac{22mL^2}{420}$	0
0	0	0	$\frac{70mLIo}{420A}$	0	0	0	0	0	$\frac{140mL\ Io}{420A}$	0	0
0	0	$\frac{13mL^2}{420}$	0	$\frac{-3mL^3}{420}$	0	0	0	$\frac{22mL^2}{420}$	0	$\frac{4mL^3}{420}$	0
0	$\frac{-13mL^2}{420}$	0	0	0	$\frac{-3mL^3}{420}$	0	$\frac{-22mL^2}{420}$	0	0	0	$\frac{4mL^3}{420}$

A = Area, m = Mass per unit length, Io= polar moment of inertia=Ix + Iy, L = Length

In general, a distributed mass matrix will give a more accurate result. However, in some circumstances the distributed mass approach may not converge. In this case use the lumped mass approach. If in doubt, use both methods and compare the results between the two.

Capacity

The absolute maximum capacities for Multiframe are as follows

Number of joints	No limit
Number of members	No limit
Number of restraints and prescribed displacements	No limit
Number of springs	No limit
Number of load cases	500
Number of joint loads	No limit
Number of member loads	No limit
Number of thermal loads	No limit
Number of members connected at one joint	18

In practice, some of the above limits may be reduced by the amount of memory available at the time the program is running. You can increase memory by modifying virtual memory settings if required.

The amount of memory required is independent of the order in which the joints are numbered. Multiframe will automatically optimise the internal numbering of the joints to best use the memory available.

The actual size of the structure you will be able to solve will depend on the number of load cases and the geometric configuration of the structure. The more load cases you use the smaller structure you will be able to analyse. If you do not have enough memory to analyse a structure, try setting up the different load cases as separate files and analysing each load file separately.

Nonlinear Analysis

Nonlinear analysis considers second order elastic nonlinearities which are due to the P-δ, P-Δ, and flexural shortening effects. The nonlinear analysis also accounts for the influence of any tension or compression only members within a structure.

P-delta effect

The implementation of the P-delta (P-δ) effect is based on the derivation presented by Ghali and Neville in which the deflected shape, member stiffness and fixed end forces are derived using stability functions. For UDL's applied over the entire length of a member, Multiframe computes the correct nonlinear fixed end forces. For other member loadings, Multiframe uses the linear fixed end forces. This has little effect for members with small axial forces and the accuracy of an analysis can be improved by simply subdividing members with high axial forces.

P-Delta effect

In Multiframe, the P-Delta (P-Δ) effect is accounted for by updating the nodal coordinates within each iteration.

Axial Shortening

Multiframe can calculate the axial shortening of a member due to flexure of the member. This nonlinear effect may be included in an analysis by selecting the appropriate option in the Nonlinear Analysis dialog. The implementation in Multiframe only considers shortening of the member due to the moments acting at the ends of the member. No account is made for the effect of loads applied to the member.

Tension and Compression only members

Tension only and compression only members can be considered by a nonlinear analyses. These types of members are removed or reinstated to the structure at the end of each iteration based upon the axial deformation of the member.

Tension and compression only members are not considered in static linear, dynamic or time history analyses.

Analysis

Nonlinear analysis in Multiframe is performed using a Newton-Raphson solution scheme. The user may control the solution process by adjusting the number of load increments, the maximum and minimum number of iterations, the convergence tolerance and the type of convergence norm.

Shear deflections

Shear deflections are not considered in a nonlinear analysis.

Plate Analysis

Multiframe now includes functions to perform flat shell finite element analysis (also called plate analysis in Multiframe). The plate elements included in the release are 3 noded, 6 degree of freedom per node elements. They are designed to handle both thin and thick plate structures. The formulation is compatible with the 6 degree of freedom formulation of the beam elements in Multiframe. As such plate analysis can be used in both pure plate structures or in combined plate-beam structures.

Patches and Plates

Plate elements are managed in Multiframe by way of patches. A patch is a planar collection of connected plate elements. Typically you will use a patch to model a slab, shear wall or plate field of some kind. You can change the mesh density inside a patch to increase of decrease the number of plates within the patch.
Material properties and loads are usually managed by modifying those properties of the patch. If you change the mesh within a patch, the plates in the patch will continue to inherit their properties and loads from the patch.

However it is also possible to override the properties or loads of an individual plate or plates within a patch. Changing properties of any plates directly will not affect its parent patch properties.

Patches and Beams

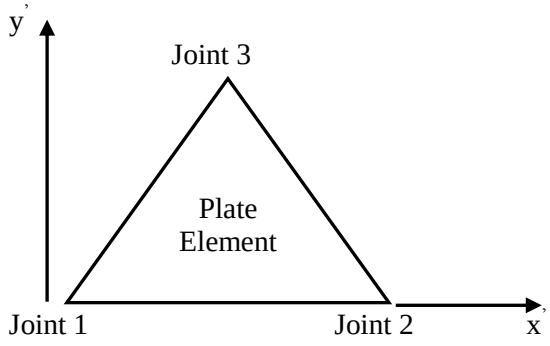
If you add a patch which has beams running along one or more of its edges, then the edge beams are assumed to be connected to the patch. As you set the density of the mesh of the patch, the edge beam(s) will be subdivided to match the internal nodes of the beam to the edge nodes of the patch. This ensures full load and moment continuity between the plate elements in the patch and the beam(s).

Recommended Workflow

We recommend that you create your beam model first, and then add patches between the beams as required. This ensures continuity between beams and patches.
Of course you are not required to create patches adjacent to beams, you can draw them standalone. In addition you can add beams to your model after you have added patches.

Conventions

Plate local coordinate system, internal forces and stress symbols are shows in the following. Z'-axis points toward to the viewer.



The plate element stresses are the forces per-unit-area that act within the volume of the element to resist the loading. These stresses are:

- In-plane direct stresses: Sxx' and Syy'
- In-plane shear stress: Sxy'

- Transverse shear stresses: Sxz' and Syz'
- Transverse direct stress: Szz' (always assumed to be zero)

The three in-plane stresses are assumed to be constant through the element thickness. The two transverse shear stresses are assumed to be constant through the thickness. The actual shear stress distribution is parabolic, being zero at the top and bottom surfaces and taking a maximum or minimum value at the mid-surface of the element.

The plate element internal forces (also called stress resultants) are the forces and moments that result from integrating the stresses over the element thickness. These internal forces are:

- Plate bending moments: Mxx' and Myy'
- Plate twisting moment: Mxy'
- Transverse shear forces: Vxz' and Vyz'

These stress resultants are forces and moments per unit of in-plane length. They are present at every point on the mid surface of the element. The sign conventions for the stresses and internal forces are illustrated in Figure below. Stresses acting on a positive face are oriented in the positive direction of the element local coordinate axes. Stresses acting on a negative face are oriented in the negative direction of the element local coordinate axes. A positive face is one whose outward normal is in the positive local x' or y' direction. Positive internal forces correspond to a state of positive stress that is constant through the thickness. Positive internal moments correspond to a state of stress that varies linearly through the thickness and is positive at the plate bottom.

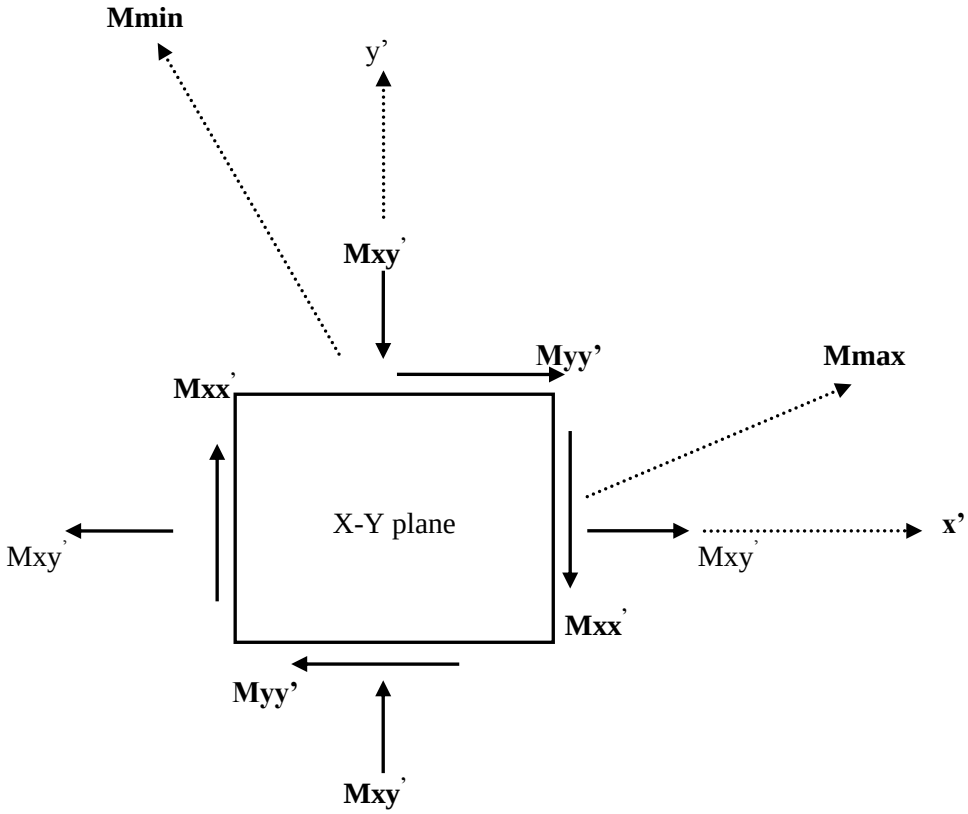
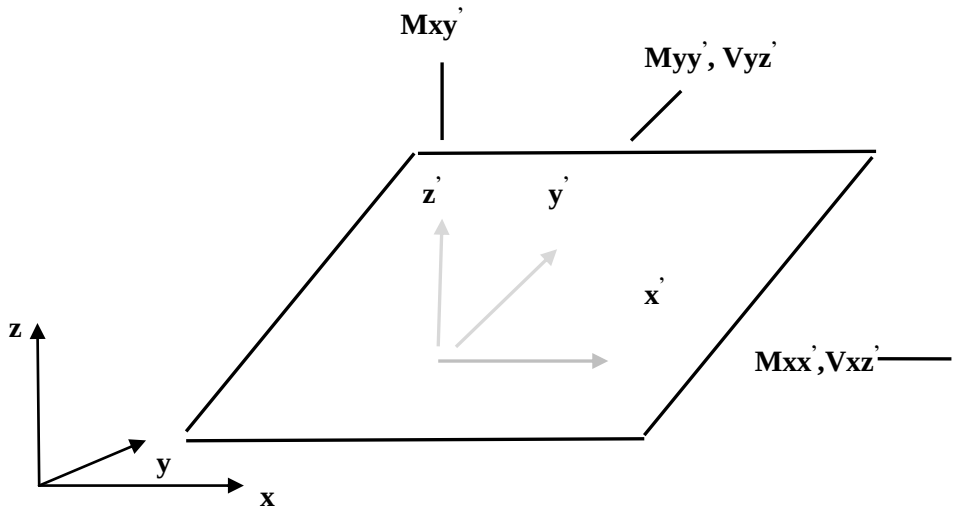


PLATE BENDING AND TWISTING MOMENTS

Transverse Shear (not shown) Positive transverse shear forces and stresses acting on positive faces point toward the viewer.

Relationships among global coordinate system, patch local coordinate system and joint positive actions are shows below:



Result Symbols

Multiframe provides structural results at all plate nodes. In plate local coordinate system, MF also gives results at Gauss points and plate centroid. Items and their symbols are described in the following table.

Item Names	Descriptions
dx', dy', dz' Φx', Φy', Φz'	Node displacements in the linear dimensions of the model. Node rotations in radians.
Vxz', Vyz'	Transverse shear forces in patch local coordinate systems.
Mxx', Myy', Mxy' Mmax, Mmin	Plate bending moments in patch local coordinate systems. Plate bending moments in principal axes.
Fxx', Fyy', Fxy' Fmax, Fmin	Plate in-plane direct forces in patch local coordinate systems. Plate in-plane direct forces in principal axes.
Mx, My, Mz	Plate moments in global coordinate systems.
Sxx', Syy', Szz' Syz', Szx', Sxy'	Normal stresses in plate/patch local coordinate systems. Shear stresses in plate/patch local coordinate systems.
Von Mises	Von Mises stress.
Tresca	Tresca stress.

References

You may find the following books useful to refer to if you need information on the matrix stiffness method of structural analysis.

Matrix Structural Analysis
R L Sack, PWS-Kent, Boston 1989

Computer Methods of Structural Analysis
F W Beaufait, Prentice Hall, New Jersey, 1970

Matrix Methods of Structural Analysis
R K Livesley, The MacMillan Co, New York, 1964

Matrix Analysis for Structural Engineers
N Willems & W M Lucas, Prentice Hall, New Jersey, 1968

Structural Dynamics, Theory and Computation
Mario Paz, Van Nostrand, New York, 1991

Structural Dynamics, An introduction to computer methods
R R Craig, J Wiley & Sons, New York, 1981

Matrix Structural Analysis
Meek J L, McGraw Hill, New York, 1971

Theory of Matrix Structural Analysis
Przemieniecki, Dover Publication, New York, 1968

Structural Analysis: A Unified **Classical and Matrix Approach**
Ghali, A. and Neville, A.M., 3rd edition, Chapman and Hall, London,1989

Matrix Structural Analysis
McGuire, W., Gallagher, R.H. and Ziemian, R.D., 2rd edition, John Wiley and Son, New York, 2000

Appendix A Troubleshooting

This appendix describes some solutions to commonly encountered problems that may occur with Multiframe.

Troubleshooting

Most Multiframe users, at some stage, will experience an error message that indicates that some of the degrees of freedom are unrestrained or that "The solution does not make sense, please check the structure, restraints and section properties." Why does this happen and what should you do if it does?

During analysis, Multiframe checks to see if there is a zero on the diagonal of the stiffness matrix. If so, it will display a error message saying either "The structure has unrestrained degrees of freedom" or "Suspected unrestrained degrees of freedom". The program will select the suspect joints in the Frame window after displaying this message. Sections 2 and 3 below describe how to deal with this problem.

After carrying out the analysis Multiframe checks the results of analysis to verify that the solution makes sense. It does this by examining the deflections and seeing if any of them are infinite. If any deflections are bad it will display the "Solution doesn't make sense" message. There are four main causes for this condition and you should check each of them if you receive the above warning message.

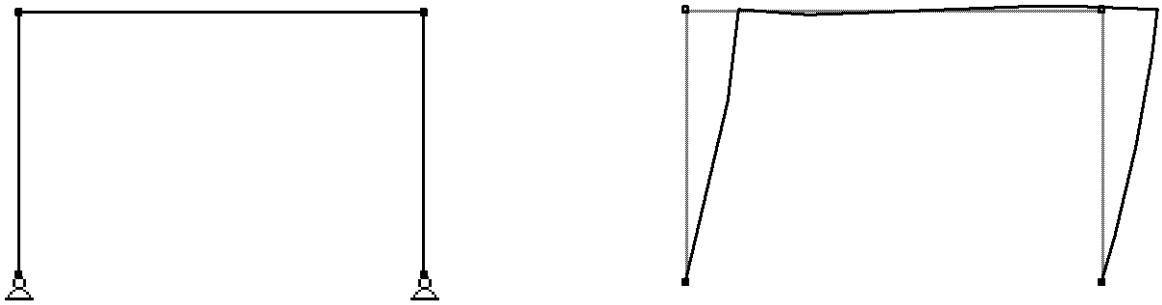
1. Check for bad section's properties

When Multiframe computes the stiffness matrix, it uses the properties in the Sections Library to determine the resistance of the sections to bending, axial and torsional deformation. If any of the key properties are zero, this will result in a zero in the matrix and consequently a bad solution. For a successful analysis, the following properties must be greater than zero:

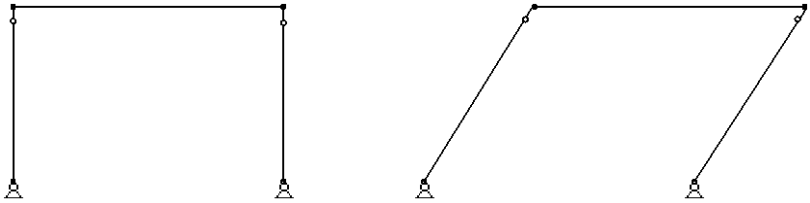
Area	Cross-sectional area of the section
Ix	Moment of inertia of the section about its strong axis
Iy	Moment of inertia of the section about its weak axis
J	St Venant torsion constant
E	Elastic or Young's Modulus
G	Shear Modulus

2. Check for mechanisms

If you use member releases (or pinned members) or pinned joints in your structure, there is a possibility that you can create a failure mechanism that allows the structure to deflect using a rigid-body mode of deformation. The simplest example is a portal frame with pinned restraint at the base of the columns under a lateral load. If all the members are rigid (no member release) analysis can proceed with no problems and the frame deforms by flexure of the columns.



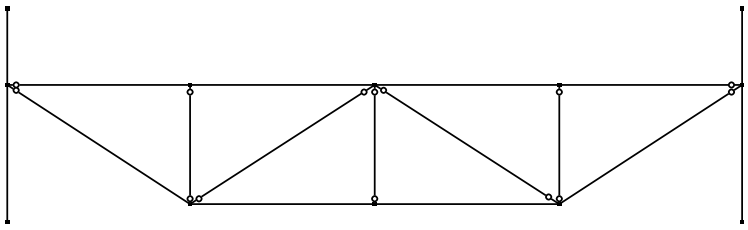
However, if we put member releases at the top of the columns, the frame is free to deflect by a mechanism allowing infinite lateral movement by rotation at the released ends of the columns. In this case, Multiframe will report the error.



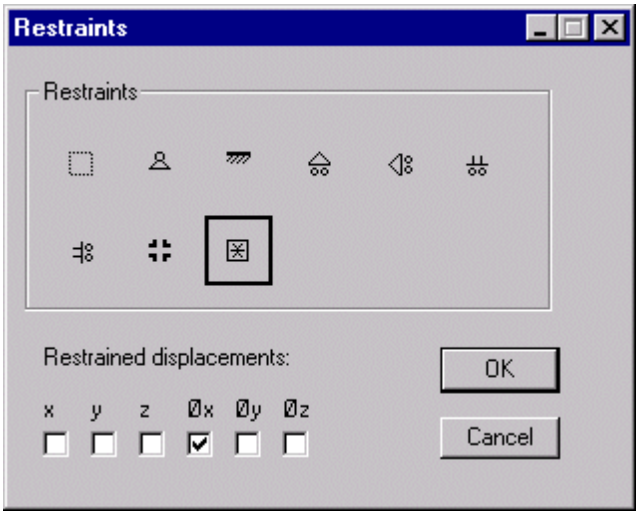
The solution is either to restrain the tops of the columns laterally or to use rigid members for the columns.

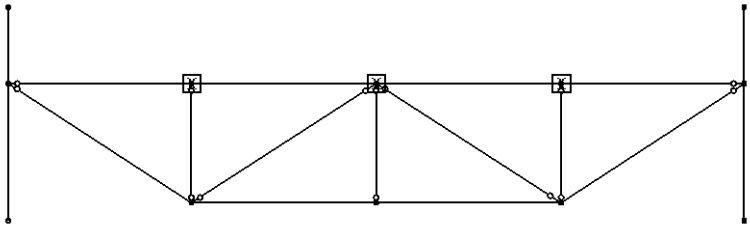
More subtle rigid body modes of deformation can develop if you release torsion (Tx') as well as bending moments (My' and Mz') when you specify the member releases. This means that it is possible to develop a torsional rigid mode of deformation in a structure. The simplest case of this is a member with torsional releases at both ends and a moment applied in the middle. In this case, there is no torsional restraint on the member and so the applied moment will cause an infinite torsional rotation.

A more complex case can develop where a group of members are connected to the rest of the structure by pin ended members and so the whole group is free to rotate to an infinite angle of rotation. A common example of this would be the top chord of a truss where the top chord is considered to be rigid but the rest of the members are pinned together.



In this case, because the top chord of the truss is pinned at its left hand end, pinned to the intermediate members along the truss, and also, pinned at its right hand end, the whole of the top chord is free to rotate about its longitudinal axis. This results in an infinite rotation of the top joints of the truss. There are three solutions to this problem. Either restrain these top joints against rotation by using a custom restraint to restrain Øx to zero





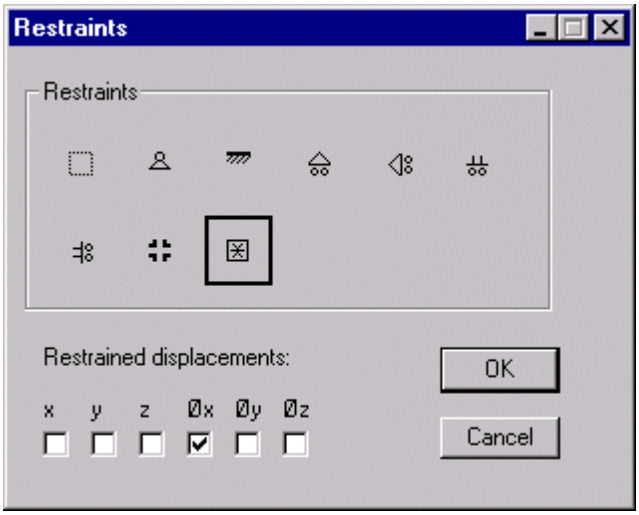
Or use a member release that does not free the torsional moment, Tx' or use the pinned joint type available in Multiframe3D version 1.6 and later. The pinned joint type sets all of the rotations of the joint to zero.

This will prevent these infinite torsional rotations without affecting the rest of the analysis.

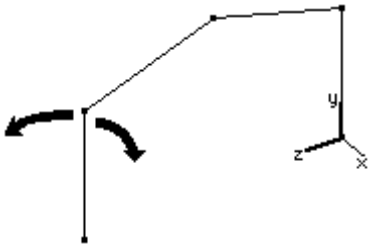
3. Check for bad restraints

When analysing a structure, it is necessary to ensure that the whole structure cannot move in a single rigid body motion. For example, if you try to analyse a two dimensional frame using Multiframe 3D, you may find the "Solution doesn't make sense" warning occurring. (This will only be a problem if the structure lies in a plane other than the Front plane. In Multiframe3D version 1.6 and later, 2D structures with 2D loading in the Front plane will be automatically recognized and analysed correctly). This is probably because you have not restrained the structure sufficiently to stop the whole frame rotating about the global x axis passing through its supports. For example, a simple or continuous beam with pinned restraints will not analyse because it is free to twist about its longitudinal axis.

The solution to these problems is to add a rotational restraint to one or more of the restraints on the structure.



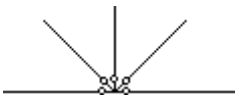
The same problem can occur with more complex 2D structures. In fact, a torsional rigid body mode of rotation will occur on any structure where the only restraints are pinned and all the restrained joints are co-linear.



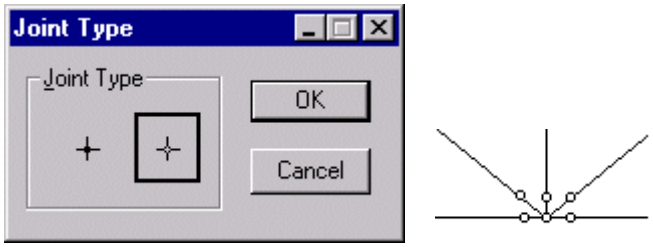
2D portal with two pinned restraints is free to rotate about the global z axis

4. Check for unrestrained joints

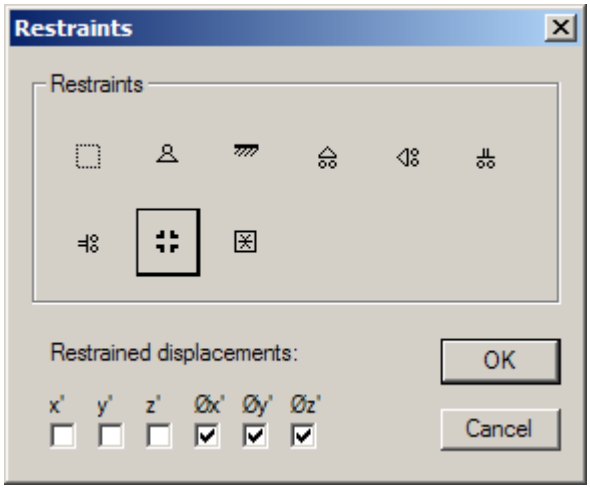
If you have a large number of members in your structure with member releases you may develop a situation where all of the members meeting at a joint have a pin at their ends. In this case there is no rotational restraint for the joint and it will result in an infinite rotation displacement of the joint.



The solution to this is to use the pinned joint type.



Alternatively, you can apply a joint restraint with only the three rotations restrained.



This will prevent the joint rotating but because the members are connected to the joint via a pinned end, they are free to deflect and rotate as usual.

Appendix B Analysing Trusses

This appendix describes how to analyse truss structures using Multiframe.

Analysing Trusses

When you create a frame for analysis by Multiframe, it initially assumes that all of the connections between the members in the structure are fully rigid, that is, there is full moment transfer across all joints. To allow you to include joints which do not transmit moment, Multiframe allows you either to use the Member Releases command to release the moments at one or both ends of a member or to use the Joint Type command to pin the ends of all of the members connected at a joint. If you are using Multiframe3D or Multiframe4D, you should use the Joint Type option. However, if you are using Multiframe2D, you will need to follow the procedures listed below.

Although you can release the end moments for as many members in a frame as you like, a problem arises if you release the moments at the ends of all of the members connected at a common joint. In this case, the joint is unrestrained against rotation, infinite rotations will be computed and the "Solution doesn't make sense" error message will appear. If you are trying to analyse a truss structure it is necessary to make all of the members in the structure pined at both ends and consequently this problem arises for all the joints in the frame.

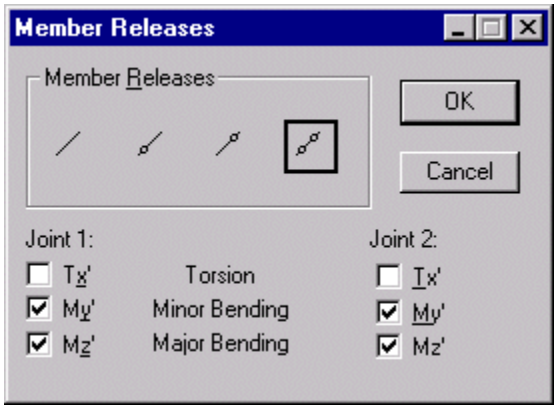
The solution to this problem is to restrain the joint against all rotations, this sets the rotations of the joint to zero, but the pins on the members allow them to deflect freely with no moment restraint.

If you are analysing a truss structure, you will probably find it easiest to follow these steps.

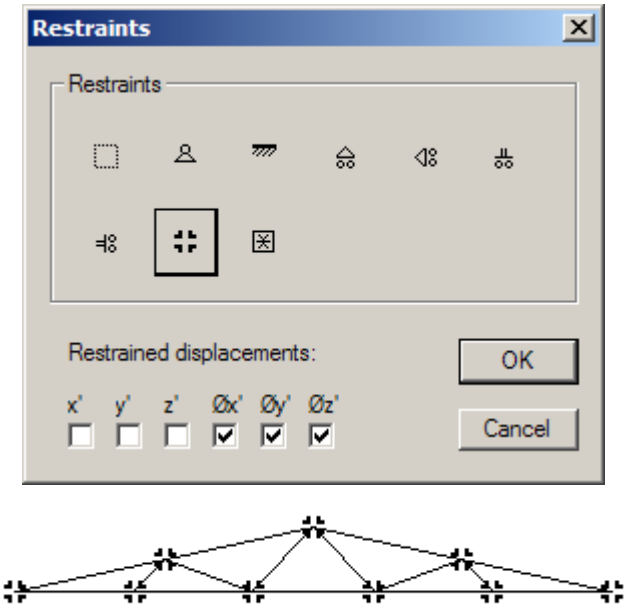
1. Draw the frame as usual

2. Use the Select All command from the Select menu to select all the members in the frame

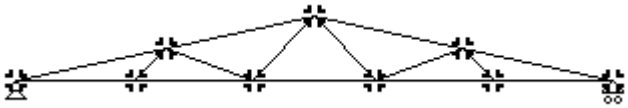
3. Use Member Releases to set all the members to Pin-Pin ends



4. Use the Joint restraint command to restrain the rotations of all of the joints in the frame to zero.



5. Use pinned restraints as usual on the foundations of the truss.



Multiframe's analysis will accurately compute the deformations of the structure due to axial forces only and the resulting rotations of all the joints will be zero.

If you find it inconvenient having so many restraints displayed in the Frame window, you can always turn them off by using the Symbols… command from the Display menu.

Appendix C Section Map File Format

This appendix describes the format of text files that may be used to map the names of sections used in Multiframe to those used by other applications. The user may select a mapping file when importing selected file formats used other applications.

Section Map Format

The format of a Multiframe section map file is described below. This section describes version 1 of the file format.

The first line of the file must contain the text “Multiframe Section Map File”.

The second line describes the file format version of the file. This line contains the text “Version” followed by an integer value. For the current file format this line should contain the text “Version 1”.

Any line after the first two lines beginning with an asterisk (*) is a comment line and is ignore when the file is read.

All other lines in the file maps a section name used within Multiframe to an alternative naming convention. Each of these lines contains up to four (4) comma or tab-delimited entries. The first two entries identify a section used within Multiframe by firstly specifying the name of a sections group and then the name of the section. The 3rd entry on this line is the name of the section used within another application. The 4th entry on the line is an additional field identifying the section, in most cases this filed will contain the name of the library used by a 3rd party application to store their sections. The 3rd and 4th entries on this line can be omitted if no section is available that corresponds to the Multiframe section

A short sample file is shown below.

```
Multiframe Section Map File
Version 1
*
* Maps Multiframe Australian section library to the Microstan Asw library

*
UB    610UB125    610UB125       Asw
UB    610UB113    610UB113       Asw
UB    610UB101    610UB101       Asw
UB    530UB92.4   530UB92.4      Asw
UB    530UB82.0   530UB82.0      Asw
```

These files can be easily generated or modified using spreadsheet applications that can save the spreadsheet in a suitable text format. Microsoft Excel allows files to be save in either tab delimited or comma delimited formats.

Appendix D Text File Format

This appendix describes a neutral text file format that can be used to send data from Multiframe to a post processor or to prepare data for Multiframe, either manually or using a pre-processor.

Text File Format

Multiframe has the capability of saving and reading files in a text format. This facility is designed to allow pre and post processing programs to transfer information to and from Multiframe. The file may also be used as a convenient summary of the data in a human readable format.

The format of a Multiframe text file is as follows.

All numbers in the file are in Fortran style format. Integers are in I6 while real numbers are in F12.4 format. Data within the file is saved in groups, each group is proceeded by the name of the group in upper case letters and followed by a number of lines with the relevant information.

The text file uses the units currently selected in Multiframe.

Job Title	Up to 80 characters describing the file and the date
METRIC or IMPERIAL Units	Number of joints
JOINTS I6 I6,F12.4,F12.4,F12.4	Start of data describing joints Number of joints Joint number, x coord, y coord, z coord …repeat line for each joint
MEMBERS I6 I6,I6,I6,I6,I6,I6,F12.4,A15,A25, I6	Start of data describing members Number of members Member no, Joint 1, Joint 2, Section Group, Section Number, Member Orientation, Group Name, Section Name, Section Shape …repeat line for each member
SECTIONS I6 A25,I6,I6,I6, F10.4,F10.4,F10.4,F10.4, F12.4,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4	Start of section data Number of different sections used in frame Section Name, Group Number, Section Number, Section Shape, Depth, Width, Web thickness, Flange thickness, Mass, Area, Ixx, Iyy, J, Young's Modulus, Shear Modulus …repeat line for each different section used in frame
RESTRAINTS I6 I6,I6,I6,I6,I6,I6,I6,I6,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4	Start of restraints data Number of restraints Restraint #, Joint #, x flag, y flag, z flag, Øx flag, Øy flag, Øz flag, x value, y value, z value, Øx value, Øy value, Øz value …repeat line above for each restraint
SPRINGS I6 I6,I6,I6,I6,I6,I6,I6,I6,F12.4,F1	Start of data for springs No of springs Spring #, Joint #, x flag, y flag, z flag, Øx flag, Øy flag, Øz

2.4, F12.4,F12.4,F12.4,F12.4	flag, x value, y value, z value, Øx value, Øy value, Øz value … repeat line above for each spring
LOADS I6	Start of loads Number of load cases
LOAD CASE A15	Start of load case Name of load case
JOINT LOAD I6 I6,I6,I6,I6,I6,I6,I6,I6,F12.4,F12.4,F12.4F12.4,F12.4,F12.4	Start of joint loads for this load case Number of joint loads in this load case Load #, Joint #, x flag, y flag, z flag, Øx flag, Øy flag, Øz flag, x value, y value, z value, Øx value, Øy value, Øz value … repeat line above for each joint load in this case
MEMBER LOAD I6 I6,I6,I6,F12.4,F12.4,F12.4,F12.4	Start of member loads for this load case Number of member loads in this load case Load #, Member #, Load Type, Left dist, right dist, left magnitude, right magnitude … repeat line above for each member load in this case
THERMAL LOAD I6 I6,I6,F12.4,F12.4,F12.4,F12.4	Start of thermal load for this load case Number of thermal loads in this load case Load #, Member #, top temp, bottom temp, thermal coeff, depth … repeat line above for each thermal load in this case …repeat for each thermal load in this case
RESULTS	Start of results of analysis
LOAD CASE A15	Start of results for load case Load case name
DISPLACEMENTS & REACTIONS I6,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4	Start of displacements and reactions Joint 3, x disp, y disp, z disp, Øx, Øy, Øz, Px react, Py react, Pz react, Mx react, My react, Mz react …repeat line for each joint
MEMBER ACTIONS I6,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4	Start of local member actions Member #, Fx1, Fy1, Fz1, Mx1, My1, Mz1, Fx2, Fy2, Fz2, Mx2, My2, Mz2 …repeat each line for each member …repeat for results for each load case
END	End of file

Following is a summary of the various flags and codes used in the text file

SECTIONS
Section Shape is an integer representing the shape of sections in the group I=1, Channel=2, Equal Angle=3, Unequal Angle =4, Tee=5, Rectangular Tube (RHS)=6, Square Tube (SHS)=7, Circular Tube=8, Circular bar=9, Rectangular bar=10, Folded Cee=11, Folded Zed=12

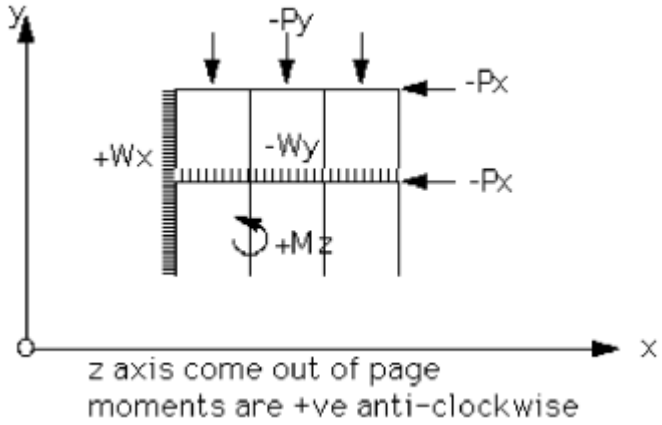
RESTRAINTS and SPRINGS
xflag, yflag, zflag, Øxflag, Øyflag, Øzflag

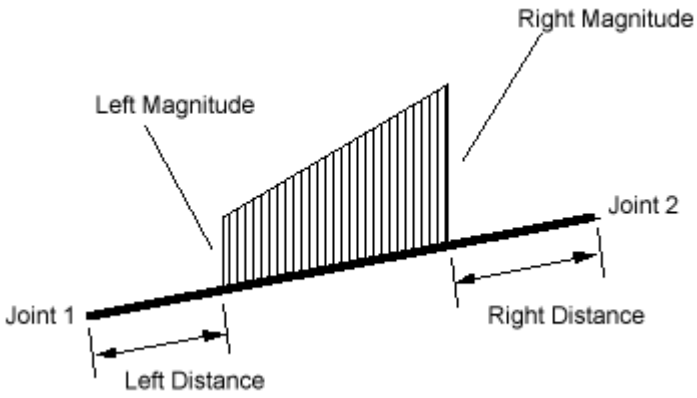
One of these should be 1 and the others zero to indicate which degree of freedom is being restrained. Only 1 degree of freedom can be set at a time for non-zero restraints.

LOADS

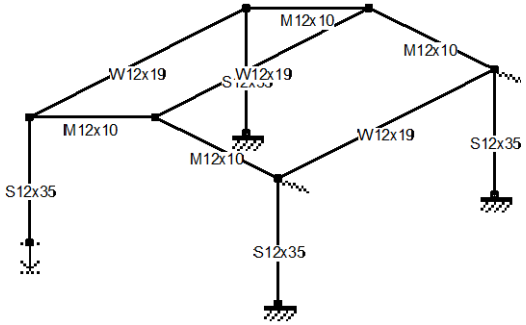
Global load codes	Local load codes
Point loads	Point loads
Px=3	Px'=-3
Py=13	Py'=-13
Pz=23	Pz'=-23
Distributed loads	Distributed loads
Wx=4	Wx'=-4
Wy=14	Wy'=-14
Wz=24	Wz'=-24
Moments	Moments
Mx=5	Tx'=-5
My=15	My'=-15
Mz=25	Mz'=-25

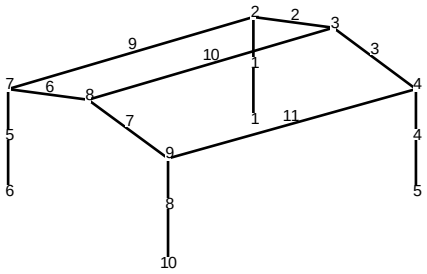
Load conventions and coordinate systems

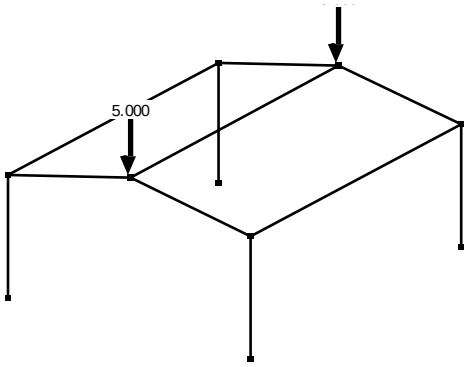




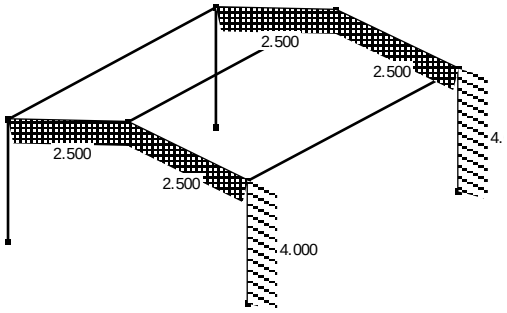
Multiframe Text Example File







TextFileFrame Load Case 1



TextFileFrame Load Case 2

```
TextFileFrame.text Fri, Apr 24, 1992
IMPERIAL
JOINTS
   10
    1      0.0000      0.0000      0.0000
    2      0.0000      9.0000      0.0000
    3     10.0000     11.1256      0.0000
    4     20.0000      9.0000      0.0000
    5     20.0000      0.0000      0.0000
    6      0.0000      0.0000     25.0000
    7      0.0000      9.0000     25.0000
    8     10.0000     11.1256     25.0000
    9     20.0000      9.0000     25.0000

   10     20.0000      0.0000     25.0000
MEMBERS
   11
    1    1    2    3   16      0.0000
    2    2    3    2    4      0.0000
    3    3    4    2    4      0.0000
    4    5    4    3   16      0.0000
    5    6    7    3   16      0.0000
    6    7    8    2    4      0.0000
    7    8    9    2    4      0.0000
    8   10    9    3   16      0.0000
    9    7    2    1  253      0.0000
   10    8    3    1  253      0.0000
   11    9    4    1  253      0.0000

SECTIONS
    3
S12x35             3   16
    114.8156     10.3007    228.9832      9.8693      1.0799  29000.6935
11150.2663
M12x10             2    4
     32.8044      2.9402     61.5955      0.9939      0.0300  29000.6935
11150.2663
W12x19             1  253
```

```
     62.3285      5.5704    129.9905      3.7597      0.1800  29000.6935
11150.2663
RESTRAINTS
    4
    1    1    1    1    1    1    1    1      0.0000      0.0000      0.0000
0.0000      0.0000      0.0000

    2    5    1    1    1    1    1    1      0.0000      0.0000      0.0000
0.0000      0.0000      0.0000
    3   10    1    1    1    1    1    1      0.0000      0.0000      0.0000
0.0000      0.0000      0.0000
    4    6    0    1    0    0    0    0      0.0000     -1.0000      0.0000
0.0000      0.0000      0.0000
SPRINGS
    2
    1    4    1    0    0    0    0    0      1.0000      0.0000      0.0000
0.0000      0.0000      0.0000

    2    9    1    0    0    0    0    0      1.0000      0.0000      0.0000
0.0000      0.0000      0.0000
LOADS
    2
LOAD CASE
Load Case 1
JOINT LOAD
    2
    1    3    0    1    0    0    0    0      0.0000     -5.0000      0.0000
0.0000      0.0000      0.0000
    2    8    0    1    0    0    0    0      0.0000     -5.0000      0.0000
0.0000      0.0000      0.0000
MEMBER LOAD
    0
THERMAL LOAD
    0
LOAD CASE

Load Case 2
JOINT LOAD
    0
MEMBER LOAD
    6
    1    2  -14      0.0000      0.0000      2.5000      2.5000
    2    3  -14      0.0000      0.0000      2.5000      2.5000
    3    6  -14      0.0000      0.0000      2.5000      2.5000
    4    7  -14      0.0000      0.0000      2.5000      2.5000
    5    4    4      0.0000      0.0000     -4.0000     -4.0000
    6    8    4      0.0000      0.0000     -4.0000     -4.0000
THERMAL LOAD
    0

RESULTS
LOAD CASE
Load Case 1
DISPLACEMENTS AND REACTIONS
    1      0.0000      0.0000      0.0000      0.0000      0.0000      0.0000
2.2274      2.5446      0.0426     -0.4825      0.0035     -9.8863
    2     -0.0338     -0.0009      0.1492      0.1749     -0.0213     -0.0015
0.0000      0.0000      0.0000      0.0000      0.0000      0.0000
    3     -0.0014     -0.1716      0.1358      0.1272     -0.0086     -0.0005
-0.0000     -0.0000     -0.0000     0.0000      0.0000      0.0000

    4      0.0309     -0.0009      0.0089      0.0011     -0.0062      0.0035
-0.0309     -0.0000     -0.0000     0.0000     -0.0000      0.0000
    5      0.0000      0.0000      0.0000      0.0000      0.0000      0.0000
-2.1622      2.4827     -0.0213     -0.1002      0.0010      9.4208
    6     -0.4120     -1.0000     -0.2251      0.1986     -0.0202     -0.0411
0.0000      1.4778      0.0000     -0.0000      0.0000     -0.0000
    7     -0.3345     -1.0005      0.1492      0.1986     -0.0202     -0.0411
0.0000     -0.0000     -0.0000     0.0000      0.0000      0.0000

    8     -0.3696     -0.8375      0.1358      0.1272     -0.0079      0.3159
-0.0000     -0.0000      0.0000     0.0000     -0.0000      0.0000
    9     -0.1930     -0.0013      0.0089      0.0011     -0.0064      0.2085
0.1930      0.0000      0.0000     -0.0000      0.0000      0.0000
```

```
   10      0.0000      0.0000      0.0000      0.0000      0.0000      0.0000
-0.2273      3.4949
-0.0213   -0.1002      0.0010    -17.6279
MEMBER ACTIONS
    1      2.5446     -2.2274      0.0426      0.0035      0.4825     -9.8863
-2.5446      2.2274     -0.0426   -0.0035     -0.8659    -10.1601

    2      2.6910      1.9920      0.0210      0.0002     -0.1149     10.1598
-2.6910     -1.9920     -0.0210   -0.0002     -0.0996     10.2054
    3      2.6723     -1.9797      0.0213      0.0005     -0.1010    -10.2023
-2.6723      1.9797     -0.0213   -0.0005     -0.1173    -10.0368
    4      2.4827      2.1622     -0.0213      0.0010      0.1002      9.4208
-2.4827     -2.1622      0.0213    -0.0010      0.0915     10.0388
    5      1.4778     -0.0000      0.0000      0.0000      0.0000     -0.0000
-1.4778      0.0000     -0.0000   -0.0000     -0.0000      0.0000

    6      0.3240      1.4794      0.0216      0.0003     -0.1196      0.0004
-0.3240     -1.4794     -0.0216   -0.0003     -0.1013     15.1241
    7      0.7492     -3.4042      0.0212      0.0005     -0.1001    -15.1272
-0.7492      3.4042     -0.0212   -0.0005     -0.1170    -19.6751
    8      3.4949      0.2273     -0.0213      0.0010      0.1002    -17.6279
-3.4949     -0.2273      0.0213    -0.0010      0.0915     19.6731
    9      0.0216     -0.0366     -0.0093      0.0004      0.1169     -0.0251
-0.0216      0.0366      0.0093    -0.0004      0.1158     -0.8899

   10     -0.0004     -0.0000     -0.0157     -0.0031      0.1971     -0.0005
0.0004      0.0000      0.0157     0.0031      0.1964     -0.0000
   11      0.0001      0.0093     -0.0092     -0.0020      0.1154      0.1163
-0.0001     -0.0093      0.0092     0.0020      0.1156      0.1163
LOAD CASE
Load Case 2
DISPLACEMENTS AND REACTIONS
    1      0.0000      0.0000      0.0000      0.0000      0.0000      0.0000
-8.1758    -23.9131     -0.0503   -0.9305     -0.0332     13.4469

    2     -0.1168      0.0086      0.1907      0.1827      0.2050      0.2610
-0.0000      0.0000      0.0000    -0.0000      0.0000      0.0000
    3     -0.2978      0.9501     -0.0073     -0.2147      0.0999     -0.0842
0.0000     -0.0000      0.0000     -0.0000     -0.0000      0.0000
    4     -0.4781      0.0094     -0.0106     -0.0015      0.0782      0.0770
0.4781     -0.0000      0.0000      0.0000      0.0000      0.0000
    5      0.0000      0.0000      0.0000      0.0000      0.0000      0.0000
43.6556    -26.0598      0.0251     0.1189     -0.0127   -149.3328

    6      6.0137     -1.0000     -0.1786      0.1959      0.2213      2.5272
0.0000    -20.0706      0.0000     0.0000     -0.0000     -0.0000
    7      1.2501     -0.9927      0.1907      0.1959      0.2213      2.5272
0.0000     -0.0000     -0.0000   -0.0000     -0.0000     -0.0000
    8      0.6043      2.0743     -0.0073     -0.2146      0.0974     -0.1494
0.0000      0.0000      0.0000     -0.0000      0.0000     -0.0000
    9      0.1750      0.0108     -0.0106     -0.0015      0.0789     -0.4912
-0.1750     -0.0000     -0.0000   -0.0000      0.0000     -0.0000

   10      0.0000      0.0000      0.0000      0.0000      0.0000      0.0000
36.2179    -29.9577      0.0251     0.1189     -0.0128    -65.0523
MEMBER ACTIONS
    1    -23.9131      8.1758     -0.0503     -0.0332      0.9305     13.4469
23.9131     -8.1758      0.0503     0.0332     -0.4782     60.1356
    2    -12.9606    -21.7098     -0.0241      0.0016      0.1299    -60.1136
12.9606     -3.8490      0.0241    -0.0016      0.1161    -31.1856
    3    -13.3874     -1.7513     -0.0253     -0.0008      0.1215     31.1849
13.3874    -23.8075      0.0253     0.0008      0.1373     81.5601

    4    -26.0598    -43.6556      0.0251     -0.0127     -0.1189   -149.3328
26.0598      7.6552     -0.0251    0.0127     -0.1072    -81.5656
    5    -20.0706     -0.0000      0.0000     -0.0000      0.0000     -0.0000
20.0706      0.0000     -0.0000    0.0000     -0.0000     -0.0000
    6     -4.1813    -19.6129     -0.0262      0.0017      0.1462     -0.0220
4.1813     -5.9459      0.0262    -0.0017      0.1217    -69.8395
    7     -6.2564      3.7271     -0.0250     -0.0008      0.1186     69.8402
6.2564    -29.2859           0.0250     0.0008      0.1365     98.9126

    8    -29.9577    -36.2179      0.0251     -0.0128     -0.1189    -65.0523
29.9577      0.2175     -0.0251    0.0128     -0.1073    -98.9071
```

```
     9     -0.0262     -0.0170      0.0122     -0.0220     -0.1434      0.0287
0.0262      0.0170     -0.0122    0.0220     -0.1606     -0.4528
    10      0.0013      0.0002      0.0187      0.0006     -0.2349      0.0031
-0.0013     -0.0002     -0.0187   -0.0006     -0.2322      0.0013
    11     -0.0002     -0.0109      0.0117      0.0055     -0.1461     -0.1365
0.0002      0.0109     -0.0117   -0.0055     -0.1468     -0.1366

END
```

Appendix E Using Spreadsheets With Multiframe

This appendix explains how you can effectively use spreadsheets with Multiframe to simplify the generation of frames and interpretation of the results of a Multiframe analysis.

Multiframe Automation

Note that you can also automatically insert data into a spreadsheet by using Multiframe's Automation capability. See the Multiframe Automation manual for more information.

Spreadsheets with Multiframe

You are probably aware that you can choose Copy from the Edit menu to copy a Multiframe picture into a word processor or drawing program. However, you may not be aware that you can also copy tables of data from Multiframe into a spreadsheet or copy and paste information from a spreadsheet into Multiframe.

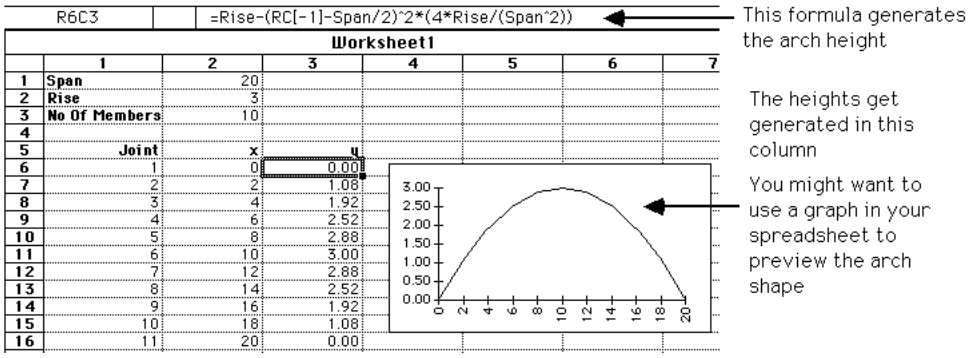
The basics of copying data in Multiframe are to select the area of the table to be copied (by dragging the mouse or shift-clicking in the usual way, see the Multiframe manual for more details) and then copy the data to the clipboard using the Copy command from the Edit menu. You can then switch to or start up your spreadsheet, click at the location where you want to paste the data, and then use the Paste command to place the data into the spreadsheet. You can copy data from any of the tables in the Data and Result windows.

To paste data into Multiframe you reverse the procedure, selecting the data to be copied in the spreadsheet, choosing the Copy command in the Edit menu, and then switching to Multiframe, selecting the area to paste it into, and then using the Paste command. You can paste data into the Joint, Member, Joint Load and Member Load tables in the Data window.

Example 1 - Generating An Arch

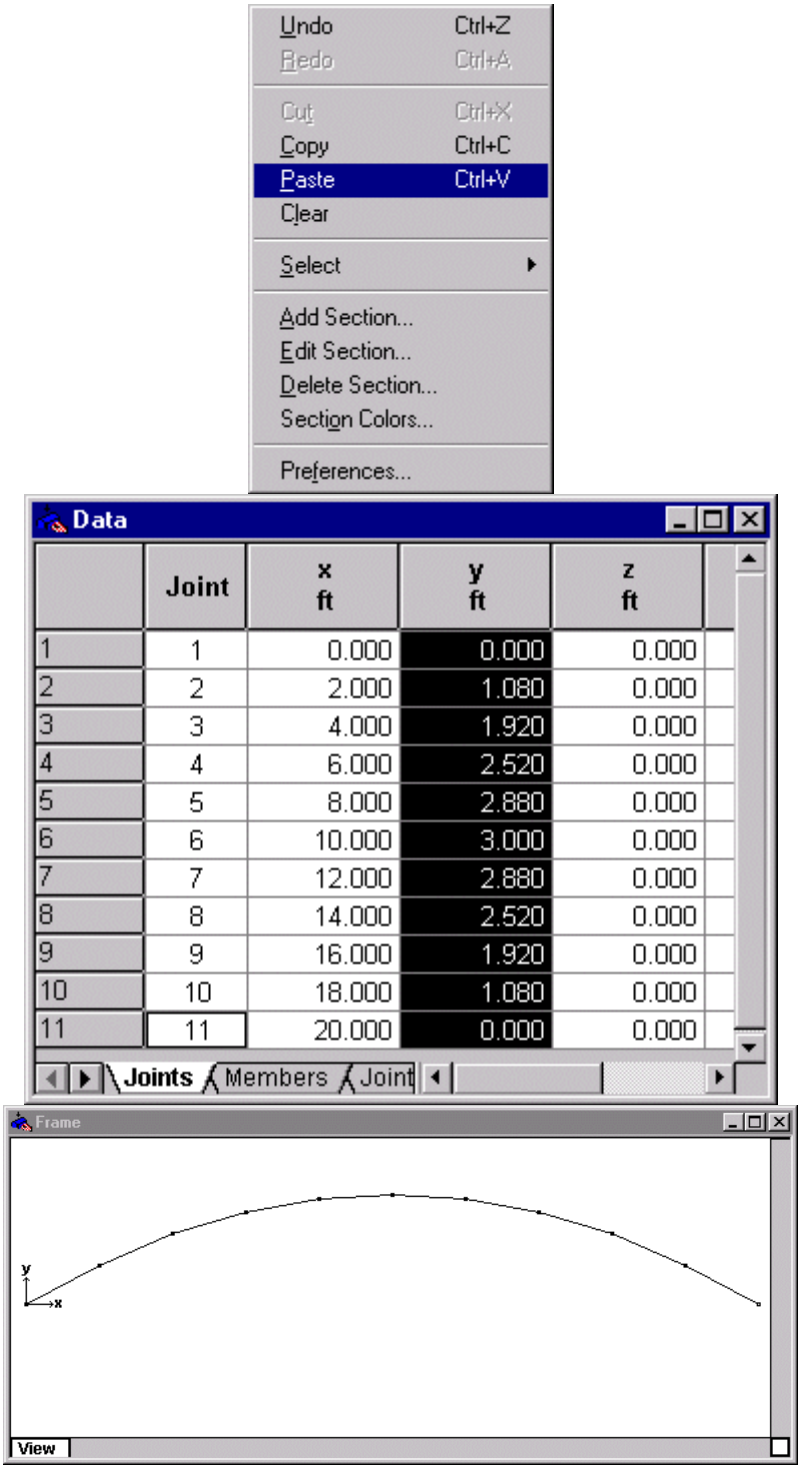
As one example of generating data in a spreadsheet and pasting it into Multiframe, consider the problem of analysing a parabolic arch. To draw or type in the geometry would be very time-consuming and difficult to modify if you wanted to consider several options for the shape of the arch. You can, however, use a spreadsheet to generate the geometry of the arch and then paste the coordinates into the Data table in Multiframe.

First, you would generate a continuous beam in Multiframe with the appropriate number of spans and then copy and paste the x column into the table in the spreadsheet. Next, write a formula for the y coordinate (or height) of the arch in the column adjacent to the data you have pasted in. You can then produce the y coordinate for each joint automatically.


R6C3
=Rise-(RC[-1]-Span/2)^2*(4*Rise/(Span^2))
This formula generates the arch height
Worksheet1
The heights get generated in this column
You might want to use a graph in your spreadsheet to preview the arch shape

	1	2	3	4	5	6	7
1	Span	20					
2	Rise	3					
3	No Of Members	10					
4							
5	Joint	x	y				
6	1	0	0.00				
7	2	2	1.08				
8	3	4	1.92				
9	4	6	2.52				
10	5	8	2.88				
11	6	10	3.00				
12	7	12	2.88				
13	8	14	2.52				
14	9	16	1.92				
15	10	18	1.08				
16	11	20	0.00				

Once you have generated the coordinates you can then copy the column of y coordinates from the spreadsheet and paste them into the y column in Multiframe.

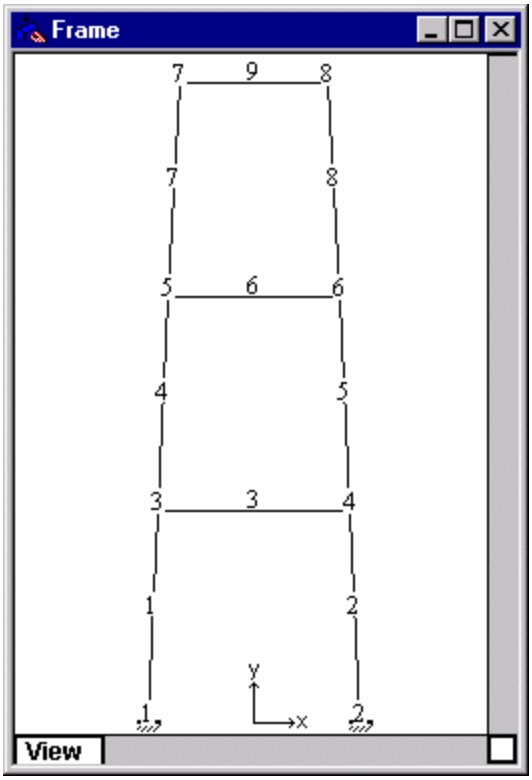


This will automatically generate the correct geometry for the arch which will be displayed in the other Multiframe windows.

If you have applied the section types, restraints and loads to the structure, these will remain intact when you paste in the new geometry, and you can immediately re-analyse to determine the results of the new shape. You can then investigate different alternatives by generating different shapes in the spreadsheet and pasting them into Multiframe. One important point to note is that pasting the coordinates in requires that the joints in Multiframe are listed in the same order as the joints in the spreadsheet. You may find the Renumber… command in Multiframe useful for ensuring that this is the case. A second point to note is that you can select just a part of a column or range of columns in Multiframe, you do not necessarily have to generate coordinates for the whole structure.

Example 2 - Generating Loads

Generating the loads for a structure is often the most time-consuming part of setting up a structural model. For example, if you have a structure subject to wind loads which depend on the height of the members above the ground, there may be hundreds of calculations required to determine the magnitudes of these loads. However, you can use a spreadsheet, in particular a spreadsheet's ability to look up values from tables, to automate the generation of loads.



If a load depends on the height of a member, you will first need to calculate the height of each member. You can do this by using the Member table from the Data window to find out which joints are at the ends of each member and the Joint table to find out the coordinates of these joints. Start by pasting these two tables into different areas of your spreadsheet and defining arrays that refer to these two tables.

JointData

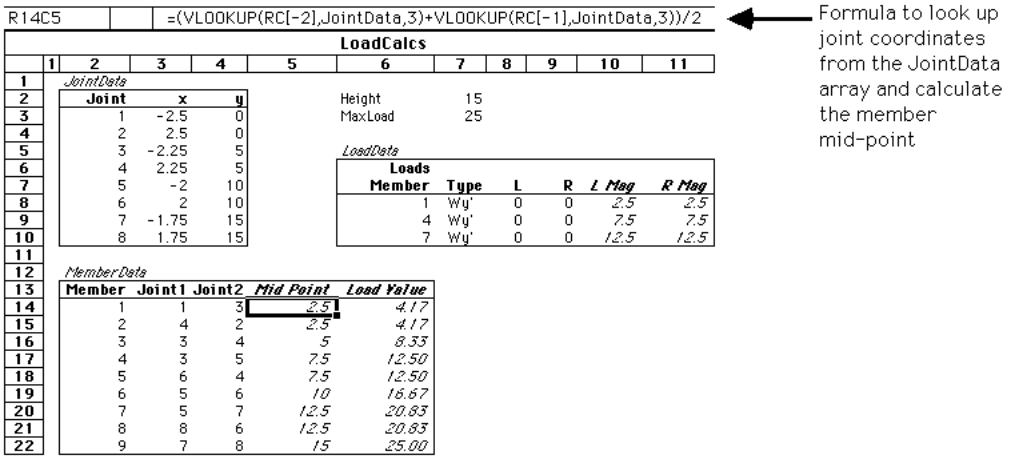
Joint	x	y
1	-2.5	0
2	2.5	0
3	-2.25	5
4	2.25	5
5	-2	10
6	2	10
7	-1.75	15
8	1.75	15

MemberData

Member	Joint1	Joint2
1	1	3
2	4	2
3	3	4
4	3	5
5	6	4
6	5	6
7	5	7
8	8	6
9	7	8

Next, add a column to the Member array and put a formula in it to look up the y coordinates of the joints and use them to find out the height of the mid-point of the member. Then add another column and put a formula in it to calculate the load magnitude from the member midpoints

(calculated values are shown in the illustrations below in italics).

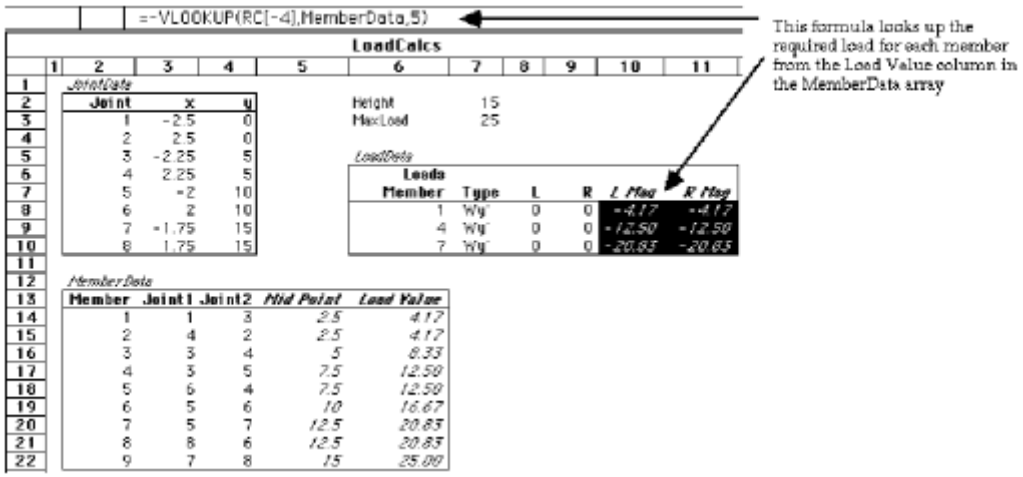

R14C5
=(VLOOKUP(RC[-2],JointData,3)+VLOOKUP(RC[-1],JointData,3))/2
Formula to look up joint coordinates from the JointData array and calculate the member mid-point
LoadCalcs
JointData
Joint x y
1 -2.5 0
2 2.5 0
3 -2.25 5
4 2.25 5
5 -2 10
6 2 10
7 -1.75 15
8 1.75 15
Height 15
MaxLoad 25
LoadData
Loads
Member Type L R L Mag R Mag
1 Wy' 0 0 2.5 2.5
4 Wy' 0 0 7.5 7.5
7 Wy' 0 0 12.5 12.5
MemberData
Member Joint1 Joint2 Mid Point Load Value
1 1 3 2.5 4.17
2 4 2 2.5 4.17
3 3 4 5 8.33
4 3 5 7.5 12.50
5 6 4 7.5 12.50
6 5 6 10 16.67
7 5 7 12.5 20.83
8 8 6 12.5 20.83
9 7 8 15 25.00

This last column of the Member Data array now contains the required load magnitude for each member in the structure. The next step is to apply the distributed load to all of the appropriate members in Multiframe and then copy and paste the Member Load table from the Data window in Multiframe into a new array in your spreadsheet.

LoadData

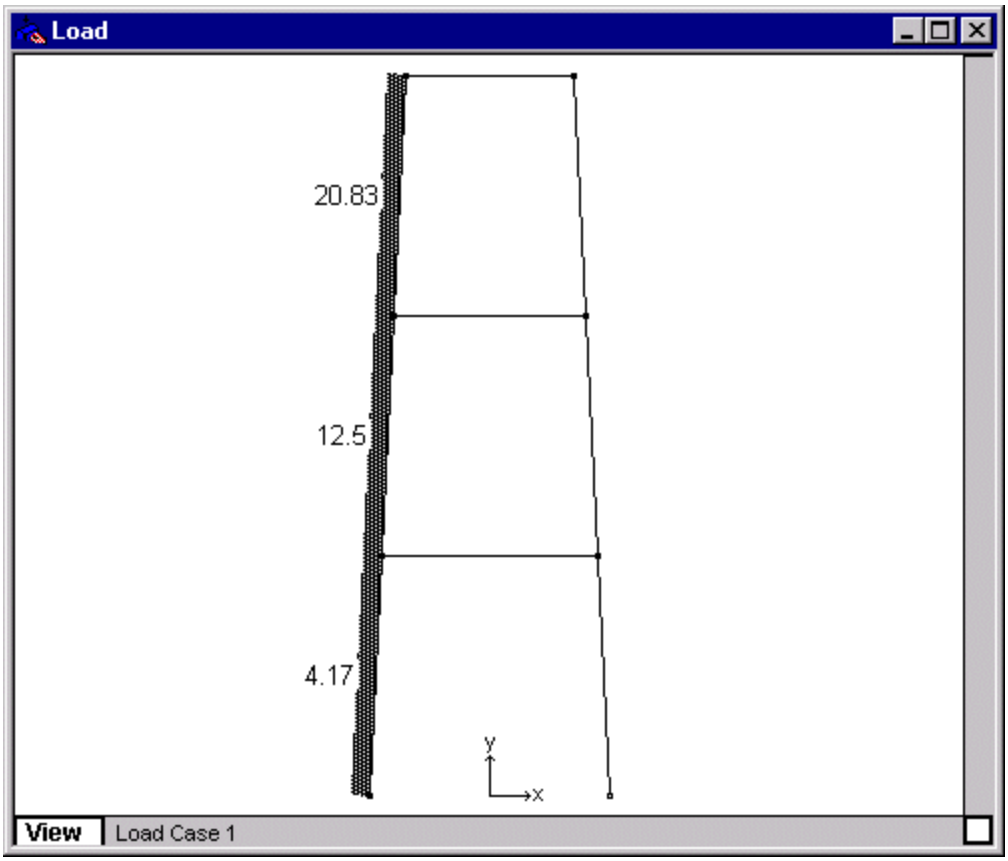
Loads Member	Type	L	R	L Mag	R Mag
1	Wy'	0	0	0	0
4	Wy'	0	0	0	0
7	Wy'	0	0	0	0

You can now put a formula for the load magnitude into the last two columns of the load array by looking up the required load magnitude that you have calculated for each member.


=-VLOOKUP(RC[-4],MemberData,5)
This formula looks up the required load for each member from the Load Value column in the MemberData array
LoadCalcs
JointData
Joint x y
1 -2.5 0
2 2.5 0
3 -2.25 5
4 2.25 5
5 -2 10
6 2 10
7 -1.75 15
8 1.75 15
Height 15
MaxLoad 25
LoadData
Loads
Member Type L R L Mag R Mag
1 Wy 0 0 -4.17 -4.17
4 Wy 0 0 -12.50 -12.50
7 Wy 0 0 -20.83 -20.83
MemberData
Member Joint1 Joint2 Mid Point Load Value
1 1 3 2.5 4.17
2 4 2 2.5 4.17
3 3 4 5 8.33
4 3 5 7.5 12.50
5 6 4 7.5 12.50
6 5 6 10 16.67
7 5 7 12.5 20.83
8 8 6 12.5 20.83
9 7 8 15 25.00

Once you have done that, you can copy and paste the load magnitudes from the load array back into the magnitude columns of the Member Load table in Multiframe.

This will update the member loads you have applied to their required values. Keep in mind the direction of these loads - you may need to use a negative sign in your spreadsheet to ensure that the magnitudes pasted into Multiframe point in the right direction.



This may sound like a rather long process but once you have set it up once you can adapt the spreadsheet for use with other structures. Also, once you get used to using the VLOOKUP command in your spreadsheet you will find it useful for many other operations (see quantities example below).

Example 3 - Calculating Quantities & Costs

When you generate a structure in Multiframe it displays a table of the sections you have used in the Sections table in the Data window.

	Section	Group	Length m	No. Used	Total Length m	Mass/L kg/m	Total Mass kg
1	TS8x8x1/2	Sq. Tube	5.006	6	30.037	72.689	2183.388
2	C8x11.5	C	4.500	1	4.500	17.112	77.004
3	C8x11.5	C	4.000	1	4.000	17.112	68.448
4	C8x11.5	C	3.500	1	3.500	17.112	59.892
5	L4x4x5/8	Angle	6.897	1	6.897	23.362	161.115
6	L4x4x5/8	Angle	6.562	1	6.562	23.362	153.304
7	L4x4x5/8	Angle	6.250	1	6.250	23.362	146.010

You can copy this table to a spreadsheet for calculations of quantities and/or costs. If you copy the whole table you can easily calculate the total weight of the structure by summing the values in the Total Weight Column.

SectionData

Section	Group	Length	No Used	Total Length	Mass	Total Mass
TS8x8x1/2	Sq. Tube	5.0	6.0	30.0	72.7	2183.4
C8x11.5	C	4.5	1.0	4.5	17.1	77.0
C8x11.5	C	4.0	1.0	4.0	17.1	68.4
C8x11.5	C	3.5	1.0	3.5	17.1	59.9
L4x4x5/8	Angle	6.9	1.0	6.9	23.4	161.1
L4x4x5/8	Angle	6.6	1.0	6.6	23.4	153.3
L4x4x5/8	Angle	6.3	1.0	6.3	23.4	146.0
				Structure Weight		*2849.2*

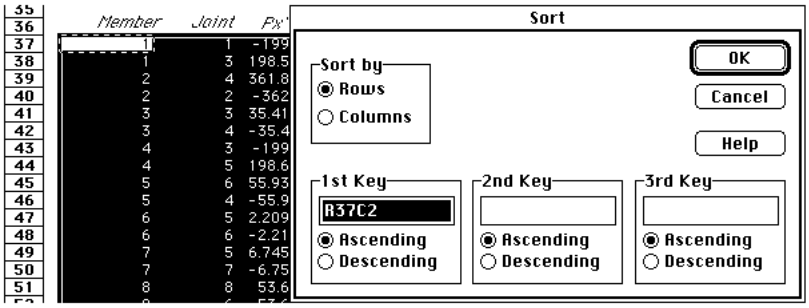
If you want to calculate the cost of structural painting you might use the values in the Length, D, B, tf and tw columns to find the total surface area and use this as a basis for your costing.

SectionData

Section	Length	D	B	tf	tw	*Surface Area*
TS8x8x1/2	30.0	203.2	203.2	12.7	12.7	*12.9*
C8x11.5	4.5	203.2	57.404	9.906	5.588	*1.0*
C8x11.5	4.0	203.2	57.404	9.906	5.588	*0.9*
C8x11.5	3.5	203.2	57.404	9.906	5.588	*0.9*
L4x4x5/8	6.9	101.6	101.6	15.875	15.875	*1.7*
L4x4x5/8	6.6	101.6	101.6	15.875	15.875	*1.6*
L4x4x5/8	6.3	101.6	101.6	15.875	15.875	*1.6*
						20.7

Example 4 - Finding Maximum Forces

After you have analysed a structure, Multiframe displays a table of the member actions in the Result window. You can copy this table to a spreadsheet and then sort it to find for example the members with the greatest axial force.



If you wish you could also add additional calculations based on the results to compute other design parameters and then sort based on the calculated values.

In Multiframe for Windows version 5.1 and later, you can also sort directly in the tables inside Multiframe by right clicking on the column heading.

Summary

Spreadsheets are an invaluable tool for all design engineers. Used by themselves they can provide quick answers to difficult design problems but used in conjunction with Multiframe they can provide you with a greatly expanded range of design tools. Next time you have a design requirement that Multiframe does not provide, consider using a spreadsheet to help provide the facilities you need to solve your design problem.

Appendix F Quality Assurance

This appendix describes the quality assurance processes used to ensure Multiframe gives reliable and accurate results.

Quality Assurance

Many Multiframe users ask us how we know that Multiframe produces the correct results. This appendix explains how Formation Design Systems has verified that Multiframe gives accurate results and what steps we take to make sure that each version of the software we ship is as reliable as possible.

Quality Principles
While it is impossible to ensure that any software product is completely free of bugs, we follow a series of engineering and testing principles and procedures to ensure that Multiframe will produce results which are consistent with the level of accuracy and thoroughness a professional engineer applies to design work. To this end we follow a development and testing path which includes use of structured programming techniques, verification of the underlying algorithms, testing of the computer implementation of those algorithms, testing of real world problems in-house and beta testing in the field at Multiframe user sites.

Structured Programming
The best defence against bugs in software is to use structured programming techniques that have been proven to improve software reliability. Without going into the technical details of our software development methodology, we summarize by saying that we utilize structured code, object oriented design, data hiding and encapsulation and fault tolerant programming practices to enhance our software's reliability. Multiframe is a complex software system of over 400,000 lines of code and we believe our history of reliability reflects the effort we have put into using reliable coding practices.

Verification of Algorithms
When new design or analysis algorithms are introduced into Multiframe, we first carry out testing on the algorithms on proven test cases with known analytical solutions. These generally come from engineering texts such as Refs 1 to 3. These test cases will include samples which independently examine the various degrees of freedom (Mx bending, My bending, Axial tension etc.) followed by examples which superimpose the effects of multiple degrees of freedom. These simple test cases are performed for structures aligned with the principal axes as well as those rotated to arbitrary angles.

Testing of Implementation
Once the basic algorithms have been proven correct, testing is then carried out on more complex sample problems to which a solution has already been established using a proven analysis program. These results may either come from structural engineering texts such as Refs 1 to 3 as well as from other results carried out by Formation Design Systems or other engineers using other software products such as SAP, Nastran etc.

Testing of Upgrades

As each new version of Multiframe is released we perform a series of tests to ensure it functions correctly. Among these tests is an analysis of a frame which exercises every different feature of Multiframe i.e. every possible member release, section type and orientation, load type, combination etc. At each release the results from this frame are compared with the results from the previous release to ensure conformance with answers which have been established as being correct.

Beta Testing
Immediately prior to the release of each new version of Multiframe, we conduct a beta test of the software. This involves sending the software to practicing engineers and having them use it on design work in progress to determine its reliability for actual design use. These beta testers provide us with feedback on the reliability and accuracy of the program as well as its useability and suitability for everyday work. Once the beta test program is completed and all testers are happy with the program, we begin shipping the commercial version.

Version Control
Each new version of Multiframe displays a version number indicating the version and the date the software was first shipped. If the version is a development, alpha test or beta test release, the version number may also include a letter and number suffix indicating the type and number of the release. A development version is usually only for internal use and is a very early demonstration of a possible new product or feature. It is highly experimental and not reliable. An alpha release is a first public release of a program for initial testing and comment, it is not reliable. A beta release is a final test version of the program released for field testing prior to commercial release. It is mostly reliable but may contain some bugs. A commercial release is a completed, debugged program reliable and ready for professional use.

For example

1.0d1	The first development release of version 1.0
1.5a2	The second alpha test release of version 1.5
1.6b2	The second beta test release of version 1.6
1.64	A commercial release of version 1.64

But we're not Perfect
We make every effort to ensure that our software will meet our users' needs and perform accurately. However, as with all complex software systems, it is possible for errors to occur. If you suspect a problem with Multiframe, please contact our technical support staff by email at support@formsys.com and explain what you believe the problem to be. In the unlikely event of a problem being found, we will correct it as soon as practicable, and send you a new corrected version of the program.

To get accurate results from Multiframe, it is necessary for you to model the problem correctly and to correctly interpret the results produced. This requires structural engineering experience combined with an understanding of matrix structural analysis. It is the users' responsibility to correctly model the structure and assume responsibility for the results.

Index

2

2D DXF....260

3

3D DXF....260

A

About Multiframe....296
Absolute Envelope....294
Actions Sub Menu....256, 257, 284, 289, 291, 292
Actions Toolbar....250, 271
Add
  Custom Section....117
  Material....122
  Standard Section....116
Add Case....154, 155, 292
Add Case Sub Menu....293
Add Connected Members....273
Add Group....278
Add Group Set....278
Add Joint....276
Add Joints....276
Add Material....122, 265
Add Member....36, 273, 274, 275
Add Section....117, 265
Add Standard Section....116, 265
Add to Group....278
Adding a combined load case....155
Adding a envelope load case....155
Adding a static load case....154
Adding Joints....93
Adding Seismic Load Cases....162
Adding Time History Load Cases....160
Advanced....276, 283
Aligning Joints....93
Analysis....183, 297
Analysis Parameters....186
Analysis Settings Table....288
Animate....284
Animations....245
Applying Dynamic Loads....161
Applying Loads....132
Arrange Icons....294
Arrow Keys....6
Automation Help....295
Automation Reference....295
Axes....266
Axial Compression....289
Axial Px'....289
Axial Shortening....186, 303
Axial Sx'....290
Axial Tension....289

B

Backspace....6
Batch Analysis....194
Bending Sby' left....290
Bending Sby' right....290
Bending Sbz' bottom....290
Bending Sbz' top....290
beta angle....113
Beta Testing....336
Bitmap Image....243, 261
Buckling Analysis....190
  Maximum Iterations....191
  Method....191
  Number of Modes....191
  Options....191
  Reference Case....191
Buckling Results....224

C

Calcs Layout....295
CalcSheet....225, 295
CalcSheet Variables....226
CalcSheet Window....247
Calculate....294
Calculating Quantities & Costs....332
Calculation Sheet....225
Calculations....225
Cancelling....184
capacities for Multiframe....302
Cascade....294
Check For Updates....296
Choosing the Time Step....163
Clear....262
Click....5
Clip
  Grey....268
  Invisible....268
  To Frame....268
  To Group....268
  To Selection....268
  To Window....268
  To Zone....268
Clip Grey....268
Clip Invisible....268
Clip To Frame....14, 268

Clip To Group....268
Clip To Selection....14, 268
Clip To Window....268
Clip To Zone....268
Clipping....12, 266, 268
  Save Clipping Zone....268
Clipping and Masking....12
Clipping Toolbar....254, 255, 270
Clipping Zones....15
Close....258
Colour....267
Colour printing....228
Comb Sx' + Sby' left....290
Comb Sx' + Sby' right....291
Comb Sx' + Sbz' bottom....290
Comb Sx' + Sbz' top....290
Combined load case
  Adding....155
Combined Load Case....293
Compressive Sx'....290
Connections....74
continuous beam....84
Convergence Tolerance....186
Convert Member to Arc....78, 274
Coordinates....101
Copy....261
Create Design Member....278
Creating a Structure....29
Ctrl key....6
Ctrl-Click....6
curved member....85
Custom Section....117
Customize Plot....212, 285
Cut....261
Cylindrical Coordinates....89

D

Data....295
Data Exchange....230
Data Sub Menu....284, 286
Data Window....247
Day Star Text....260
Daystar Text Files
  Export....240
Deflection....284
Delete....6
  Section....119
Delete Joint Masses....112
Delete Case....157, 293
Delete Group....278
Delete Group Set....278
Delete Material....265
Delete Member....80, 220, 274
Delete Section....119, 265
Deleting a load case....157
Depth....33
Design Member
  Create....278
  Remove....278
Disconnect Members....276
Disconnecting Members....96
Displacements Table....288
Display Menu....283
Distributed....193
double click....6
double precision....184
Drag....6
Drawing....29
Drawing Depth....33, 266
Drawing Settings....35, 276
Drawing Toolbar....248, 271
Duplicate....87, 275
Duplicate current load case....155
DXF....231, 259
  Export....239
Dynamic Line Constraints....34, 277
Dynamic Load....282

E

Edit
  Section....119
  Zones....268
Edit Case....156, 292
Edit Group....278
Edit Group Set....278
Edit Load Library....158, 262
Edit material....265
Edit Menu....261
Edit Section....119, 265
Editing a load case....156
Editing Coordinates....101
Editing Layout....295
Editing Load Cases....156
Editing Loads Numerically....150
Editing Restraints Numerically....107
El Centro....158
elastic critical load....190
elastic stiffness matrix....190
End Spring Actions....201
End Springs....129
Envelope load case
  Adding....155
Envelope Load Case....293
Evaluating Expressions....181
Exit....259
Export....258
  2D DXF....260
  3D DXF....260

Bitmap Image....243, 261
Day Star Text....260
Daystar Text Files....240
DXF....239
Microstran Archive....260
Multiframe Text File....239
Multiframe v7....242, 261
Navisworks....244
NavisWorks....261
SDNF....242
SDNF Text....261
Space Gass Text....260
Space Gass Text Files....241
Spreadsheet Text....240
Text File....260
Text Files....260
Time History Results Text Files....242
Time History ResultsText File....261
VRML....239, 260
Export Sub menu....259
ExportMicrostran Archive Files....240
Extrude....99, 275

F

Factored Load Case....154
File Export....239
File Import....230
File Menu....257
File Toolbar....248, 270
Fixed Time Interval....160
Font....266
force diagram....214
Format Toolbar....270
Formatting Toolbar....252
Frame....295
Frame Menu....278
Frame Window....247

G

Generate....81, 82, 275
Generate Toolbar....248, 271
Generating a continuous beam....84
Generating a curved member....85
Generating a multi-story frame....82
Generating a portal frame....81
Generating a regular frame....87
Generating An Arch....327
Generating Loads....330
Geometry Menu....273
Geometry Toolbar....249, 271
Global Dist'd Load....281, 282, 283
Global Distributed Load....135, 138, 139, 142, 143, 144
Global Joint Load....133, 281
Global Joint Moment....281
Global Moment....147, 281
Global Mx....289
Global My....289
Global Mz....289
Global Point Load....145, 281
Graphic....244
Grid....30, 266
Group
Add....278
Add to....278
Delete....278
Edit....278
Remove from....278
Group Menu....278
Group Set
Add....278
Delete....278
Edit....278
Group Sets....107
Group Toolbar....253, 270
Grouping....107
Groups....109

H

Help Menu....295
Help, Multiframe....295
Home....7

I

Import....258
Microstran Archive Files....236
SDNF....238
Space Gass Text Files....237
Import Sub Menu....259
Importing Load Data....159
Increments....186
Installing Multiframe....3
Intersect Members....79, 276
Intrinsic Functions....181
Inverse Iteration....191
iterations....186
Izmir....158

J

Jacobi method....191, 192
Joint....281
Labels....279
Linking....279
Mass....278
Masses....112
Numbers....103
Orientation....125
Orientation....279

Restraint 278
Restraints 105
Spring 278
Type 124, 279
Joint and Member Numbers 103
joint coordinates 92
Joint Displacement 196, 224
Joint Labels 104, 279
Joint Linking 279
Joint Load 133
Global Joint Moment 281
Local Joint Load 281
Local Joint Moment 281
Prescribed Displacement 281
Unload Joint 281, 282, 283
Joint Loads Table 286
Joint Mass 278
Joint Masses 112
Joint Masses Table 287
Joint Moment 133
Joint Orientation 125
Joint Orientation 279
Joint Reactions 196, 223
Joint Restraint 105, 278
Joint Spring 106, 278
Joint Toolbar 250, 271
Joint Type 124, 279
Joint Variables 182
Joints
Linking 110
Joints Linking 110
Joints Table 286

K

Keyboard techniques 6
Kobe 158

L

Learn Multiframe 296
Learning Multiframe 3
Legends 24
Licensing 264
Preferences 264
Linear Analysis 184
Linear Results 293
Linked Joints Table 286
Linking Joints 110
Selecting 111
Load 295
Global Distributed 135, 138, 139
Global Point 145
Joint Force 133
Joint Moment 133
Local Distributed 137, 138, 141, 219
Local Moment 148
Prescribed Displacement 134
Self Weight 151
Thermal Load 149
Load case
Add Combined 293
Add Envelope 293
Add Seismic 293
Add Static 293
Add Time History 293
Load Case 293
Load Case Toolbar 251, 271
Load Cases 154
Load Cases Table 287
Load Library 158
Load Menu 281
Load Toolbar 251, 272
Load Window 247
Loads
Editing 150
Global Moment 147
Local Point 146
local and global axes 297
Local Dist'd Load 281, 282, 283
Local Distributed Load 137, 138, 141, 219
Local Joint Load 133, 281
Local Joint Moment 281
Local Moment 148, 282
Local Point Load 146, 282
Lumped 193

M

Mask Grey 269
Mask Invisible 269
Mask Out Selection 16, 269
Mask To Frame 269
Mask To Selection 269
Mask To Selection 16
Mask To Window 269
Masking 16, 266
Masking Sub Menu 269
Mass Matrix Type 193
Master-Slave 110
Materials Sub Menu 265
Matrix Stiffness Method 297
Maximum Actions 198
Maximum Envelope 294
Maximum Forces 333
Maximum Member Actions Table 288
Maximum Member Stresses Table 288
Maximum Stresses 199
Member 115
Component Type 131, 279
End Releases 126

End Springs....129, 279
Labels....279, 280
Masses....279
Numbers....103
Offsets....279
Orientation....279
Releases....279
Rigid Offsets....128
Shear Area....131, 279
Type....279
Types....127
Member Actions....187, 197, 201, 299
Member Actions Table....288
Member Buckling....203, 288
Member Component....279
Member Details....199
Member Details Table....288
Member Diagrams....215, 218
Member End Properties Table....286
Member End Springs....279
Member End Springs Table....286
Member Geometry....102
Member Geometry Table....286, 287, 288
Member Labels....104, 279, 280
Member Load
Global Dist'd Load....281, 282, 283
Global Moment....281
Global Point Load....281
Local Dist'd Load....281, 282, 283
Local Moment....282
Local Point Load....282
Member Self Weight....282
Thermal Load....282
Unload Member....281
Member Loads Table....286
Member Masses....112, 279
Member Material....121
Member Modlling....131
Member Offsets....128, 279
Member Orientation....113, 279
Member Properties Table....286
Member Releases....126, 279
Member Self Weight....152, 282
Member Shear Area....279
Member Stresses....198
Member Stresses Table....288
Member Toolbar....249, 271
Member Type....279
Member Types....127
Member Variables....182
memory requirements....184
Merge Close Joints....276
Merge Members....79, 276
Microstran Archive....259, 260
Microstran Archive Files
Export....240
Import....236
Minimum Envelope....294
Mirror....99, 275
Modal Analysis....192
Analysis Method....192
Convergence....192
Maximum Iterations....193
Mode Shape Scaling....193
No of Modes....192
Modal Results....224
Mode Shape....293
Moment My'....289
Moment Mz'....256, 257, 289, 291, 292
moment of inertia....299
Mouse techniques....5
Move....36, 94, 275
Moving a joint....91
Moving a member....95
Multicore....264
Multicore Optimisation Analysis....264
Multiframe load codes....321
Multiframe Structure....230
Multiframe Text....236, 259
Multiframe Text Example File....322
Multiframe Text File....260
Multiframe Text File
Export....239
Multiframe v7....261
Export....242
Multiple load cases....154

N

Natural Frequencies....203
Natural Frequencies Table....289
Navisworks....244
NavisWorks....261
New....258
No Clipping....268
No Masking....269
Nonlinear Analysis....184, 303, 304
Nonlinear Results....188, 203, 293
Number Format....265

O

Online Support....296
Open....258
Open Library....258
Open Sections Library....258
Orientation
Joint....125
origin....36

Osaka 158

P

Page Setup.....228, 258
Pan 10, 266
Participating Mass Ratio 203
Participation Factors 203
Paste 262
P-delta effect 185, 303
P-Delta effect 186, 303, 304, 307
Pinned Joint.....125
Plot 285, 295
Plot Colour 285
Plot Window 206, 247
Plots
  Envelope Cases 222
Plotting Envelope Cases 222
portal frame 81
Precision.....184
Preferences.....200, 262
Prescribed Displacement.....134, 281
Prescribed Displacements Table 286
Print Diagrams 229, 258
Print selected members 229
Print Summary 228, 258
Print Window.....230, 258
Printing.....228
Printing Diagrams 229
Printing Reports 228
Properties 259, 265

Q

Quality Assurance.....335
Quality Principles.....335

R

Rayleigh Damping Factors 163
Reactions.....207
Reactions Table.....288
Re-analyse.....251
Redo 261
References.....308
regular frame 87
Remove Design Member 278
Remove from Group 278
Render 16, 218, 284
Render Toolbar 253
Rendered length 17
Rendering Toolbar 270
Renumber.....103, 276
Reorder Cases 157, 292
Report.....295
Report Layout 295
Report Window.....247
Reporting
  Font.....263
  Preferences.....263
  Repeat Table Headers 263
  Shade Alternate Rows.....263
  Using Microsoft Word.....263
Rescale.....97, 275
Resizing a Member 97
Restraints 104
Restraints Table 286
Result 295
Result Layout.....295
Result Window 195, 247
Results 195
  Buckling.....203
  Natural Frequencies 203
  Time History.....203
  Viewing 195
Results Sub Menu 284, 288
ResultsNonlinear.....188, 203
Reverse Members 276
Rotate.....98, 275
Rotating a 3D View 8

S

Save 258
Save As 258
Saving Calculations 228
SDNF
  Export 242
  Import 238
SDNF Text.....259, 261
Section 115
  Deleting 119
Section Colours.....265
Section Map Format 317
Section Orientation 113
Section Properties 115
Section Properties Variables.....227
Section scale 17
Section Sub Menu.....265
Section Type 115, 120, 279
Sections
  Editing 119
Sections Library.....115, 117, 123
Sections Table.....120, 287
Seismic Load Case.....162, 293
Select
  Member Actions 272
Select 262
  All 272
  Horizontal 272
  Joint 272
  Joint Labels.....272

Member....272
Member Labels....272
Section....272
Sloping....272
Vertica....272
Select
Joint....272
Select
Joint Labels....272
Select
Joint....273
Select
Joint Labels....273
Select
Joint....273
Select
Joint Labels....273
Select
Groups....273
Select
From Window....273
Select Design
Member Labels....273
Select Design Member....273
Select Menu....272
Selecting Joints....75
Selecting Members....75
Self Weight....151
Self Weight Load Case....293
Self Weight Loads....153
Semi-rigid connection....129
Setting up the printer....228
Shear....100, 275
Shear Area....131
Shear deflections....303
Shear Sy'....290
Shear Sz'....290
Shear Vy'....289
Shear Vz'....289
Shift or Ctrl keys....6
Shift-Click....6
Shrink....11, 266
Sign Convention....297
single precision....184
Size....29, 266
Size To Fit....12, 266
Size To Fit Sub menu....267
Snapping....33
Snapping to Joints....34
Snapping to Members....34
Sorting load cases....157
Space Gass Text....259, 260
Space Gass Text Files....237
Export....241
Spherical Coordinates....91
Spreadsheet Text
Export....240
Spreadsheets....327
Spring Actions....201
Spring Member Actions Table....288, 289
Spring Member Stiffness....280
Spring Member Table....287
Springs....106
Springs Table....286
Standard Section....116
Static load case
Adding....154
Static Load Case....293
Status Bar....267
Steel Designer Help....295
Step Loads Table....287
stiffness matrix....298
stress diagram....214
Stresses....214
Stresses Sub Menu....284, 290
Structural Grid....30
Structural Grids....266
Structure Diagrams....207
Subdivide Member....76, 274
subspace iteration....192
Symbols....23, 206, 283
Symbols Toolbar....252, 254, 255

T

Tab....6
Table Data....245
Tables
Analysis Settings....288
Joint Loads....286
Joint Masses....287
Joints....286
Linked Joints....286
Load Cases....287
Member End Properties....286
Member End Springs....286
Member Geometry....286, 287, 288
Member Loads....286
Member Properties....286
Prescribed Displacements....286
Restraints....286
Results....288
Sections....287
Spring Member s....287
Springs....286
Step Loads....287
Thermal Loads....286
Tensile Sx'....290

Text File Format 319
Text Files 260
Thermal Load.....149, 282
Thermal Loads Table 286
Tile Horizontal 294
Tile Vertical 294
Time History (Ctrl + H) 294
Time History Analysis 193
Time History Envelope Cases.....205
Time History Load Case 160, 293
Time History Results 203
Time History Results Text Files
Export.....242
Time History ResultsText File.....261
Time Menu.....294
Toolbar.....267
Actions 250
Clipping 254, 255
Drawing 248
File 248
Formatting.....252
Generate 248
Geometry 249
Group 253
Joint.....250
Load 251
Load Case.....251
Member.....249
Render.....253
Symbols 252, 254, 255
View.....250
View3D.....253
Windows 253
Toolbars 270
Torque Tx'.....289
Troubleshooting 309
Trusses 315
Turkey.....158

U

Undo 261
Units.....267
Unload Joint..... 133, 138, 139, 142, 143, 145, 281, 282, 283
Unload Member 281
user defined material.....122

V

Variable Time 160
View
Preferences.....263
View Menu 265, 266
View Sub Menu 266, 269
View Toolbar 250, 270
View3D Toolbar 253, 270
Viewing Applied Time History Loads.....161
Views 7
Visible.....30
VRML.....260
Export 239

W

Window Menu 294
Window Toolbar 270
Windows Toolbar 253

Z

Zones
Edit.....268
Save Clipping Zone 268
Zoom.....8, 266